ORIGINAL MANUAL
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COMPUTER-AIDED DRAFTING
STUDENT MANUAL

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1015 North Sixth Street
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MATCCAD™
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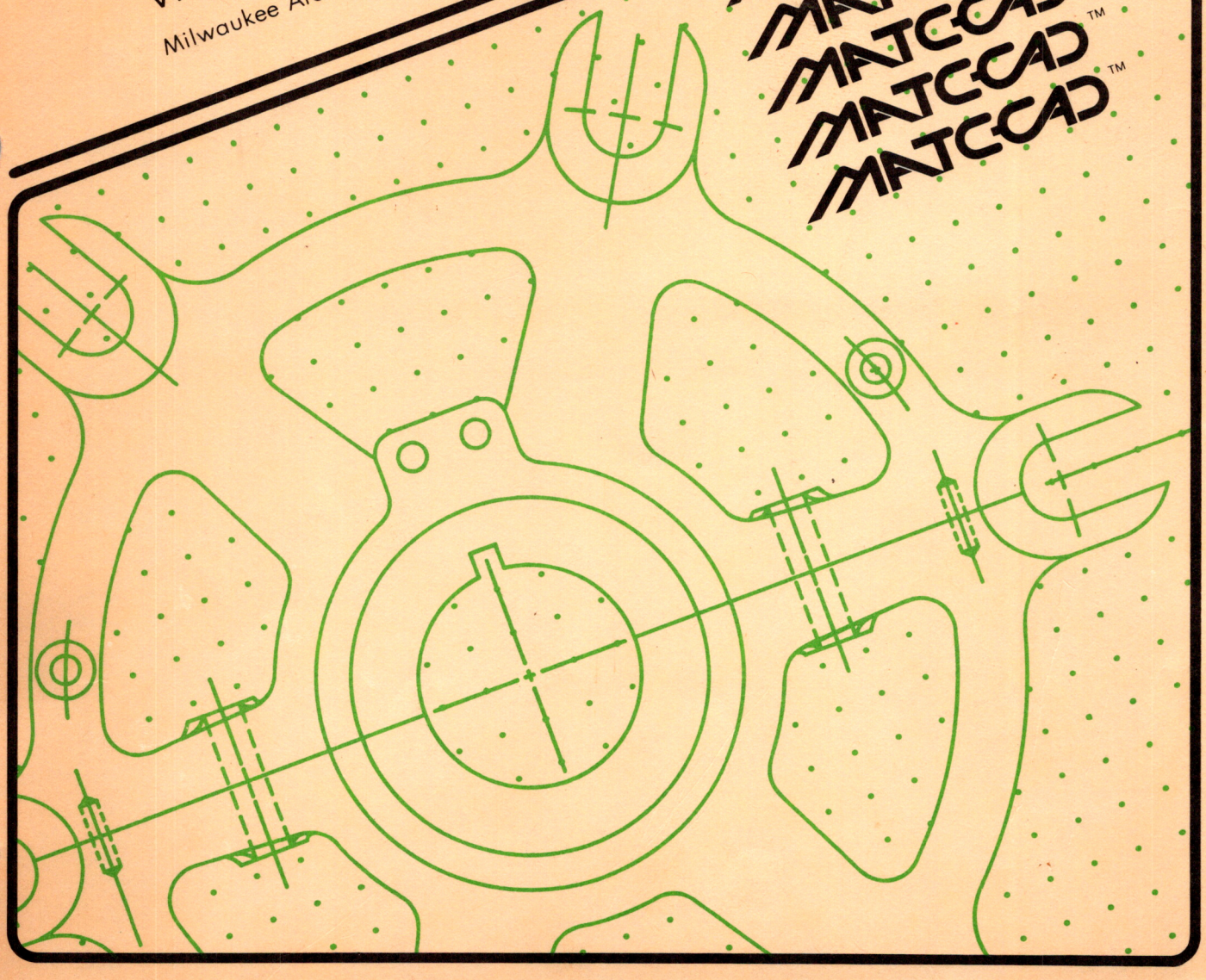

NOTE: THE DIGITIZER SHOULD BE PUT IN SLOT 5. THIS IS NEW.

MORE INFORMATION TO FOLLOW.

NEW MATC CAD Log-In Procedure NEW

1. Insert MATC CAD Disk 1 and Disk 2 into the disk drives.
2. Turn Apple computer on. (Switch is at left back corner)
3. Turn display screen on.
4. Wait 20 seconds for text to appear on screen.

	SCREEN MESSAGE	USER RESPONSE
5.	WELCOME CAD 1, TO APPLE II PASCAL 1.1	X (no RETURN)
6.	EXECUTE WHAT FILE ?	#5:MATC ↵
		(Wait 40 seconds)
7.	MATC CAD COMPUTER-AIDED DRAFTING MILWAUKEE AREA TECHNICAL COLLEGE	↑ Rn
8.	n>	CADDS ↵
9.	OLDPART = O, NEWPART = N, REPLY =	N (no RETURN)
10.	Part Name:	(enter part name, RETURN)
11.	#	(Enter commands to create part)

MATC CAD Filing and LOG-Out Procedure

1.	#	EXIT ↵
	FILE = F, DON'T = D, REPLY =	F (no RETURN)
2.	INSERT YOUR PART DISK IN DRIVE 1 THEN PRESS ANY KEY TO CONTINUE	(insert part disk, press RETURN key)
3.	PART NAME:	(enter part name, press RETURN key)
4.	REPLACE MATC CAD DISK1 IN DRIVE 1, THEN PRESS ANY KEY TO CONTINUE.	(replace disk, press RETURN key)
5.	NEW PART = N, OLD PART = O, GOODBYE = G, REPLY =	G (no RETURN)
6.	n>	LOGOUT ↵
7.	**TASK n TERMINATED**	(remove MATC CAD disks, turn off computer, turn off display)

Jan 30 1984

MATC CAD Computer-Aided Drafting
Milwaukee Area Technical College

NEW PLOTTING INSTRUCTIONS FOR REVISION 011184

1. Insert PLOTTER DISK in Drive #1
   MATC CAD DISK 2 in Drive #2

2. Turn power on and procede through the normal LOG-IN and calling up your part to be plotted.

3. The image on the screen can now be positioned to the scale and position desired with a combination of #ZOOM and #PAG DIS commands.

   Note that #ZOOM SCLn command has not been implemented yet.

4. The #SEL PLOT DSCLn) command can be used if desired to automatically scale dimensions and text to a more appropriate scale. See page 5-3 in student manual.

5. The plotter to be used and paper size must be selected next. When the command #SEL PLOT) is entered, a list of currently supported plotters and paper sizes will list out on the screen. Select the appropriate # (no RETURN) and you are ready to plot.

6. Insert Paper and pen into you plotter, and when ready enter the command #PLOT).

Please Note: If you do not select a plotter to be used with the #SEL PLOT command, and then type in #PLOT) the system will crash with a value range error.

Feb 6 1984

Aligning the Apple Graphics Tablet and MATC CAD Menu Overlay

Bring up the MATC CAD software, get into CADDS.

Press the ESC key twice to get to full screen graphics.

Aligning the Plastic Overlay on the Apple Graphics Tablet

While holding the digitizing pen vertically, move the pen from side to side, and up and down at the lower left and bottom right corners of the graphics tablet until the cursor (flashing cross) just disappears from the screen at at each lower corner.

The plastic overlay should then be moved around until the two pen positions you have just located are as close as possible to the two 1/16 inch circles at the lower corners of the plastic overlay. Once you have located the plastic overlay in the correct position, tape it down securely.

Aligning the MATC CAD Menu

Repeat the above procedure to find the pen positions on the graphics tablet where the cursor just disappears from the upper left and upper right corners of the screen. These pen positions locate the lower left and lower right corners of the MATC CAD menu overlay. Once the menu has been aligned, tape it down securely.

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matc
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MATCCAD™

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FOREWORD

MATCCAD™

This manual is designed to provide a computer graphics operator with essential information about computer graphics, practice exercises to prepare for design station activity, and numerous hands-on exercises. The goal is to gain sufficient skill to construct two-dimensional objects while inputting the construction geometry into computer memory (data base) and retrieving the information for use in design, drafting, and/or production.

The prerequisite background for using the manual is one year of drafting experience or a course in drafting, blueprint reading, or technical illustration. It is very helpful to have some math such as algebra and geometry to understand coordinate geometry. A background in computers or computer programming is not necessary.

The manual is divided into **13 units** consisting of the **introduction** (statement of purpose), **behavioral objectives** (expected student performance), **reference section, assignment section,** and **lab exercises.** The manual introduces the student to the use of a new tool in drafting and design. The exercises were selected to develop specified 2-D geometric skills and drawing data base with a language based on Computervision Corporation's Computer Aided Drafting and Design System (CADDS) ™. Specific occupational area applications are excluded to maintain a generic course with emphasis on geometry, except for the final project, which is chosen by the student. Further occupational application, 3-D, and other computer capabilities are included in advanced courses. The degree of success depends first on the knowledge to perform drafting, design, and engineering tasks, and then on use of computer graphics as a tool to enhance productivity. Students successfully completing the course have sufficient experience to be as productive as employees with six months of on-the-job training.

ACKNOWLEDGMENTS

MATCCAD™

Joseph Cleveland, Manager of Operation and Design Support Engineering Section of the General Electric Medical Systems Division, experienced a severe shortage of CAD/CAM operators in 1978 and began to assist MATC in developing a training program, which started in 1980. An advisory committee was established in 1979 consisting of industry and education members who guided the development of computer graphics and its integration in each of 11 occupational areas. A generic computer graphics operator course was suggested as a background, and specific occupational applications are to be included in other courses.

Dennis Shanahan of General Electric, **Jerry Lange** of MATC, **Max Raisor** of Brigham Young University in Provo, Utah, and **Jack Stahl** of General Dynamics Corporation in San Diego, California, provided the initial guidance in development and organization of curriculum and laboratory instruction. The first instructors, **John Erbes** (also manual editor), **Larry Andersen,** Mechanical Design Technician Department, and **Grant Andersen,** then CAD Manager at Bucyrus-Erie Company, established the first course materials. In the summer of 1981, **Richard Tupta,** Welding Technology Department, developed the units with rationale, objectives, and exercises, based on the lecture notes of **Larry Andersen. Charles Evans,** Electrical Technology Department; **Arnold Fogelberg,** Civil Engineering Technology; **George Robak,** Architectural Technology; **Wilmer Adomat,** Numerical Control; the late **Joe Niesl,** Commercial Art; **David Baule,** Telecasting; **Dwight Freund,** Project Staff; **Mike Stanislawski,** Numerical Control; **Darwin Hocking,** Electrical Technology; and **James Stencel** and **Jon Simons** of the Louis Allis Company, part-time instructors, authored sections or advised on development of the manual. Course instructors in monthly meetings, assisted by **Dr. Spiro Mehail,** Curriculum Coordinator, reviewed tests, units, student evaluations, follow-up studies, and student logistics analyses, which led to this edition. Preliminary units have been tested on over 1,000 students using a Computervision Corporation CAD system. **Computervision Corporation** is acknowledged for its outstanding technical support and interest in industry-education partnerships to provide up-to-date skills for today's technical students and tomorrow's high tech employees. Finally, **John Erbes,** MATC Computer Graphics Coordinator, is acknowledged for his editorial leadership in compiling all Computervision hardware and software information along with instructional materials and research to create this manual.

Dr. Dwight Freund, while on leave from the University of Wisconsin — Waukesha, conducted a feasibility study on the practicality of emulating the features of the Computervision system on a microcomputer. Dr. Freund, assisted by **Jim Chang** and **Henry Drydyk,** demonstrated that an emulation was possible with the design and development of a prototype working system. The MATC CAD team completed the design to maintain the features and speed of the Computervision system. The complex emulation package necessitated a sharing of responsibilities with specialized contributions as follows:

Jim Chang and **Henry Drydyk,** program architecture, command interpreter, I/O routines, data base, menu, dimensioning, and layering; **Victor Ren,** coordinate systems, geometry creation, manipulation routines, analytical mathematics, and screen control features; **Robert Kennedy,** supervision, plotting, and splines; **Caroline Mensink,** text, data base, and code optimization; **John Erbes,** system documentation, hardware, and project coordination.

The MATC CAD software is truly an outstanding achievement, with each contributor deserving of special recognition. Congratulations!

The two pilot test sites with **Edmund Paul** of Waukesha County Technical Institute and **Robert Moore** of Lakeshore Technical Institute and their students are acknowledged for patience and assistance during the test of the prerelease version.

Ralph Schubert, vice-president of Computervision Corporation, assisted MATC by providing an industrial system to emulate, authorized MATC the use of the Computervision command syntax, and encouraged the development of MATC CAD.

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The combination of MATC CAD software and the student manual will provide a cost-effective alternative to training on industrial systems. Where once there was a shortage of trained high tech CAD persons, design education will soon require CAD skills.

Victor Langer
Computer Graphics Project Director

Special acknowledgment is given to the following advisory committee members guiding MATC in development:

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COMPUTER-AIDED DRAFTING

Unit 1
Fundamentals of Computer Graphics

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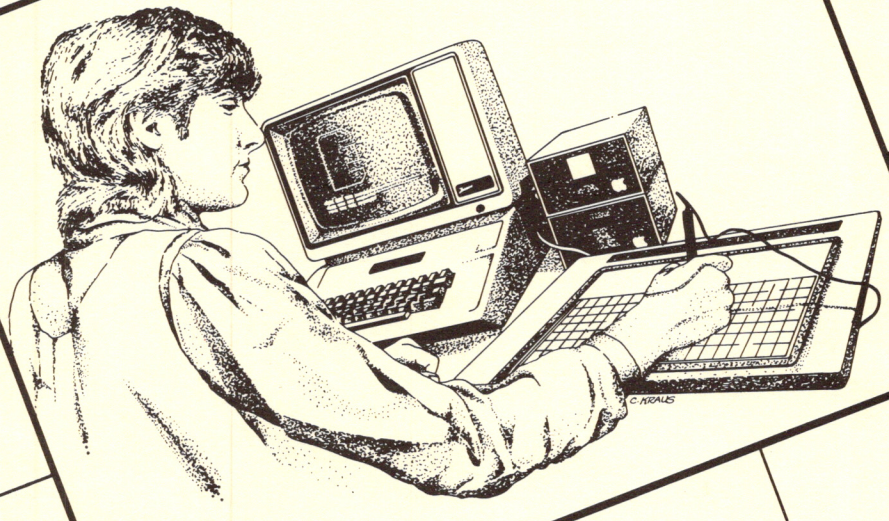

Introduction

Although computer-aided design and drafting equipment has been around since the 1960's it has only been in the last five or so years that this technology has been widely adopted by industry. More and more businesses and industries are turning to interactive computer graphics for its potential to increase productivity, increase quality, promote standardization, and provide a common data base of information for design, accounting, production, and manufacture. This introductory unit covers the application areas, equipment, advantages, and operation of interactive computer graphics.

Objectives

At the completion of this unit you will be able to:

1. Identify the parts of a computer graphics system and describe their function.
2. Identify major manufacturers and users of computer graphics systems.
3. Describe the major applications of interactive graphics.
4. Describe the advantages of computer graphics systems for typical applications as compared to traditional graphics methods.

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Part I
General Description of Computer Graphics

What is Interactive Computer Graphics?

Interactive computer graphics is a relatively new technology that gives the draftsperson, designer, engineer or artist the ability to create a graphical representation of some object or design on what could be basically described as a television-type display screen connected to a computer.

An interactive computer graphics system is one which interacts with the user during its operation. While information about the drawing being created is being added to the computer's memory, the user is presented information from the computer to continually update the operator's understanding of the level of a command.

Who Uses Interactive Computer Graphics?

At the present time interactive graphics systems can be found at nearly every large industrial or manufacturing company like Boeing, Rockwell, GE, General Motors, etc., and locally at companies like Allen-Bradley, Allis-Chalmers, A.O. Smith, Bucyrus-Erie, Louis Allis, and others.

What is Computer Graphics Used For?

Computer graphics systems are widely used for all types of graphics work, where time savings, precision, and uniformity are needed in the production of graphical displays of information. The type of work that is done on an interactive computer graphics system includes two-dimensional mechanical drafting, three-dimensional mechanical design, electrical schematic drafting and design, printed circuit board design, layouts in architectural design, facilities layout and planning, process piping, numerical control machining, numerical control welding or cutting, mapping for land use or utilities documentation, technical illustration, business graphics, and graphic arts design. Most of the systems currently being manufactured are designed for production use by industry to reduce costs and manpower requirements for producing and storing drawings. Many current systems interface directly with production manufacturing equipment to automatically perform machining and fabricating tasks that would be tedious, difficult, dangerous, or impossible for humans to do.

What Kind of Equipment is Used With Computer Graphics?

An interactive computer graphics design system is much like a typical medium-sized computer installation with a central computer (typically a 16-bit mini computer), magnetic tape and disk drives for storage, and various specialized graphics devices such as graphic display screens, plotters, and digitizers which are used to input or output information to and from the computer.

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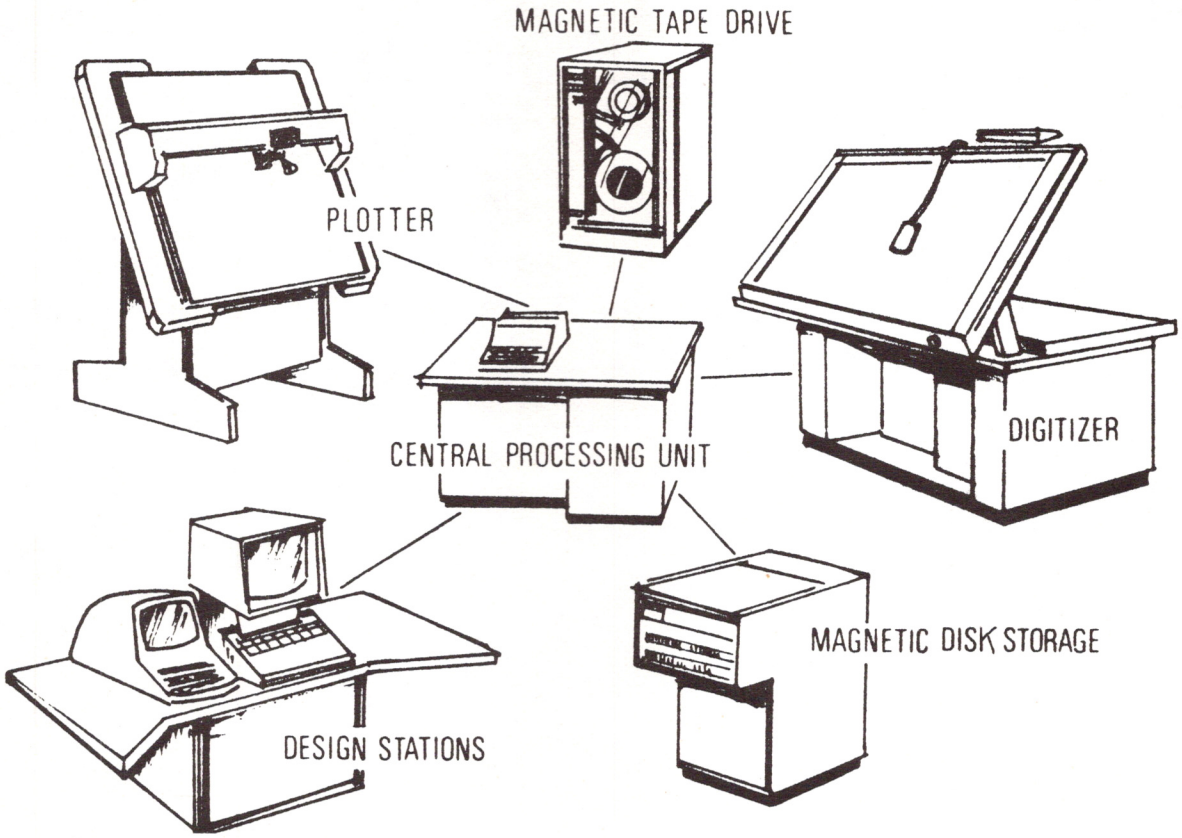

The central part of any computer system is the central processing unit (CPU). This unit processes all of the arithmetic operations and graphics commands. Hooked up to the CPU are all of the various peripheral units used for inputting commands, outputting the drawings, and for storing parts or drawings for later use. The user does all of his or her work at a graphics design station. The graphics design station typically includes a display screen for viewing the drawing, a terminal keyboard for entering graphics commands, and a digitizing tablet for position location. A plotter is used for the final output of the drawing and magnetic disk and magnetic tape drives are used to store the drawings for later use. A disk drive typically contains from 80 to 300 million characters worth of storage. 20 million characters of this space is for the operating system, the scratch file area that is used while you are working on your part, and the applications software programs. The remainder of the space is for the storage of the user's drawings.

Part II
Advantages of Computer Graphics Systems

A major advantage in using a computer graphics system over manual drafting methods arises from the need for copying of standard symbols, lines, and arcs over and over again when producing drawings with pencil and paper. A graphics system can have scores of standard symbols, drawing formats, and standard circuit configurations stored in its memory data base that can easily be called up and placed on a drawing. Symbol sizes, orientations, and locations are easily specified and changed before or after being put on a drawing.

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Minor or major drawing changes can easily be made without working with large sheets of paper, electric erasers, or various manual drafting tools. For a system properly set up with trained operators, drawings can be made 2 to 20 times faster than with conventional manual drafting techniques.

These basic drafting advantages are recognized by industry, with computer graphics systems widely used in electrical and mechanical design applications. As costs come down and the popularity of these systems increases, many new applications will arise. At present because of the high costs associated with the purchase of a graphics system, use is restricted to industries where cost effectiveness is an important criterion for implementation. In the competitive world of business these advantages are of utmost value and include:

1. Increased productivity. The number of drawings produced in a given time period is significantly higher with computer-aided drafting.

2. Reduced error. The error rate is reduced because of the ability of most systems to easily and in some cases automatically verify that the part geometry being created is correct.

3. Reduced personnel requirements. Because of the increase in productivity, a company can take on more work without increasing personnel.

4. Improved turn-around time for drawing changes. Changes in design can be easily and quickly implemented with less effect on production.

5. Improved drawing quality. Drawings are easier to read and interpret by engineering, production, and manufacturing, due to standard forms for linestyles, text and dimensions.

6. Improved standardization. Drawing standards can be easily developed and maintained.

7. Lower costs. Most of the above advantages result in lower costs to the company.

8. Compatibility with computer aided manufacturing. Many computer graphics systems can produce cutting tool paths that can be used with numerically controlled manufacturing equipment.

Part III
Graphic System Components and Operation

Graphic Display Technologies

There are two main types of graphics display devices used in interactive computer graphics equipment: the vector display and the raster display. The raster display operates in the same way as a television set does by generating many horizontal scan lines from the top of the screen to the bottom. A vector display screen operates much like an oscilloscope where a line is drawn directly from one screen position to another. Decreases in memory costs have made the raster scan display the preferred choice for most design stations.

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What Devices are Used for Graphic Input?

Most of the input devices provide the option of entering commands and data through a typewriter keyboard or a digitizing tablet. The size of the tablet varies, from 11 x 11 inches up to 4 x 6 feet. The operator uses a digitizing pen with the tablet much in the same way a draftsperson uses a paper and pencil. The digitizing pen identifies a specific point on the screen by means of a "cursor" on the screen. The cursor is a lighted mark on the screen that continually indicates where the pen is positioned on the tablet. After a command is entered in to the system such as to insert a circle or a line, the digitizing pen is used to locate where the circle or line is to be placed. Typically a pen will have a spring loaded switch in the tip which when pressed against the tablet, enters the current coordinates of the pen position into the system. On most systems a software option known as a menu provides for the ability to insert most of the commands by means of the pen and tablet. To do this, a certain area of the tablet is reserved for a set of function keys or menu. The computer is programmed to recognize these function keys as specific commands when the pen is pressed down in the menu area.
A system may instead be equipped with a joystick or two thumbwheels, one to move the cursor right and left, and the other to move it up and down. Once the cursor has been moved to the desired location, a digitizing button is pushed to select that location.

What Devices are used for Graphic Output?

The hard copy output device on a computer graphics system may be an electro-mechanical plotter, electrostatic plotter, or a photoplotter. The electro-mechanical plotter is reminiscent of a standard modern drafting table, with a large flat surface. It is a relatively slow operating device, but with the use of good quality pens and drafting paper, a high quality drawing suitable for publication is produced.

The electrostatic plotter produces a drawing of quality almost as good as that of an electromechanical plotter but considerably faster, especially for large, complex drawings. The electrostatic plotter operates on the same principle as a Xerox copying machine by putting a charge on the paper, which attracts particles of black toner which are then fused to the paper surface with heat. The photoplotter is a specialized plotter that is used to expose photographic film that is used typically for the production of printed circuit boards or integrated circuits.

A graphics system can also produce paper tape or magnetic tape that can be fed into computer-controlled machines used in manufacturing and fabrication.

Who Manages the Graphics System?

The computer graphics system is under the supervision of a system manager. The system manager starts the system, loads the appropriate software from the library of tapes, and clears the system for use of the graphic work stations. After the operator has been notified that the system is ready for use, he or she then logs into the system. It is the system manager's duty to prepare back-up magnetic tapes of the parts created, and to report hardware problems and software bugs.

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How Does the System Operate?

The operation of an interactive graphics system can be described at either the external or internal level of operation.

Externally, you as an operator use the graphics system by giving to the computer the appropriate commands needed to create and modify graphical entities such as points, lines, circles, arcs, etc. These commands are given in a straight-forward English-like language which means that you do not have to know anything about how a computer works to operate the system. You enter a command and the result is displayed on the screen. Most work is done at a grahics design station, which typically has a graphics display screen for viewing the part, a computer keyboard type terminal for entering the graphics commands, and a graphics digitizing tablet with digitizing pen for locating positions or selecting existing entities from your drawing. Other computer grahics systems might use light pens, joysticks, x-y position thumbwheels, dual graphics display screens, or other devices for ease of operation. After the operator has created a part or drawing it can be stored for later use or recall on a magnetic disk drive or magnetic tape.

Internally, the computer goes through a complex sequence of operations while an operator is working on a drawing. As the operator types in a graphics command, the computer accesses from the magnetic disk the appropriate computer program that performs that command. Then the entity that is created (line, circle, etc.) is added to the long list of entities that describes the makeup of your drawing. Finally, the entity you have just created is displayed on the display screen and added to the picture image representation of your drawing. An important concept to understand is that the computer does not store a picture of your drawing. What is created is a file containing a brief description of each of the entities that makes up your drawing. This data base can then be used as a general and common source of information for other operations such as bills of material, computer-aided manufacturing (CAM), report generation, group technology, etc.

A computer graphics system can be described from two viewpoints, that of hardware such as the central processing unit, magnetic disk and tape drives, design station terminals and plotters, or from the viewpoint of software. The software on a graphics system can be divided into three categories. First is the operating system software. This is the collection of programs that run the computer resources available to the user. Secondly there is the applications software consisting of up to several thousand programs that when executed perform the various graphics commands. Last, there is the collection of all of the drawings or parts that are created using the graphics system.

Who Are the Major Manufacturers of Industrial CAD/CAM Graphics Systems?

Manufacturer	Primary Applications
Computervision	Mechanical Design and Drafting Electrical Design and Drafting Process Piping Numerical Controlled Machining Mapping

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GE Calma	Mechanical Design and Drafting Mapping Manufacturing Applications Integrated Circuit Design
Applicon	Mechanical and Electrical Design and Drafting
Auto-Trol	Mechanical Design Drafting
Gerber	Printed Circuit Artwork Mechanical Design and Drafting
Intergraph	Plant Design Numerical Control Electronic Design Piping Mechanical Design Architectural Mapping
IBM CADAM	2-dimensional Mechanical Drafting

The MATC CAD Computer-Aided Drafting System

The MATC CAD Computer-Aided Drafting software developed by Milwaukee Area Technical College has been designed to provide a low-cost introduction to the field of interactive computer graphics.
The MATC CAD software runs on a standard Apple IIe or Apple II Plus with 64K of memory, two flexible disk drives, and an Apple Graphics Tablet. The software has been designed to simulate the general two-dimensional drafting features of Computervision Corporation's CADDS3 (TM) system exactly.

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COMPUTER-AIDED DRAFTING

Unit 2
Computer Graphics System Operation

MATCCAD™

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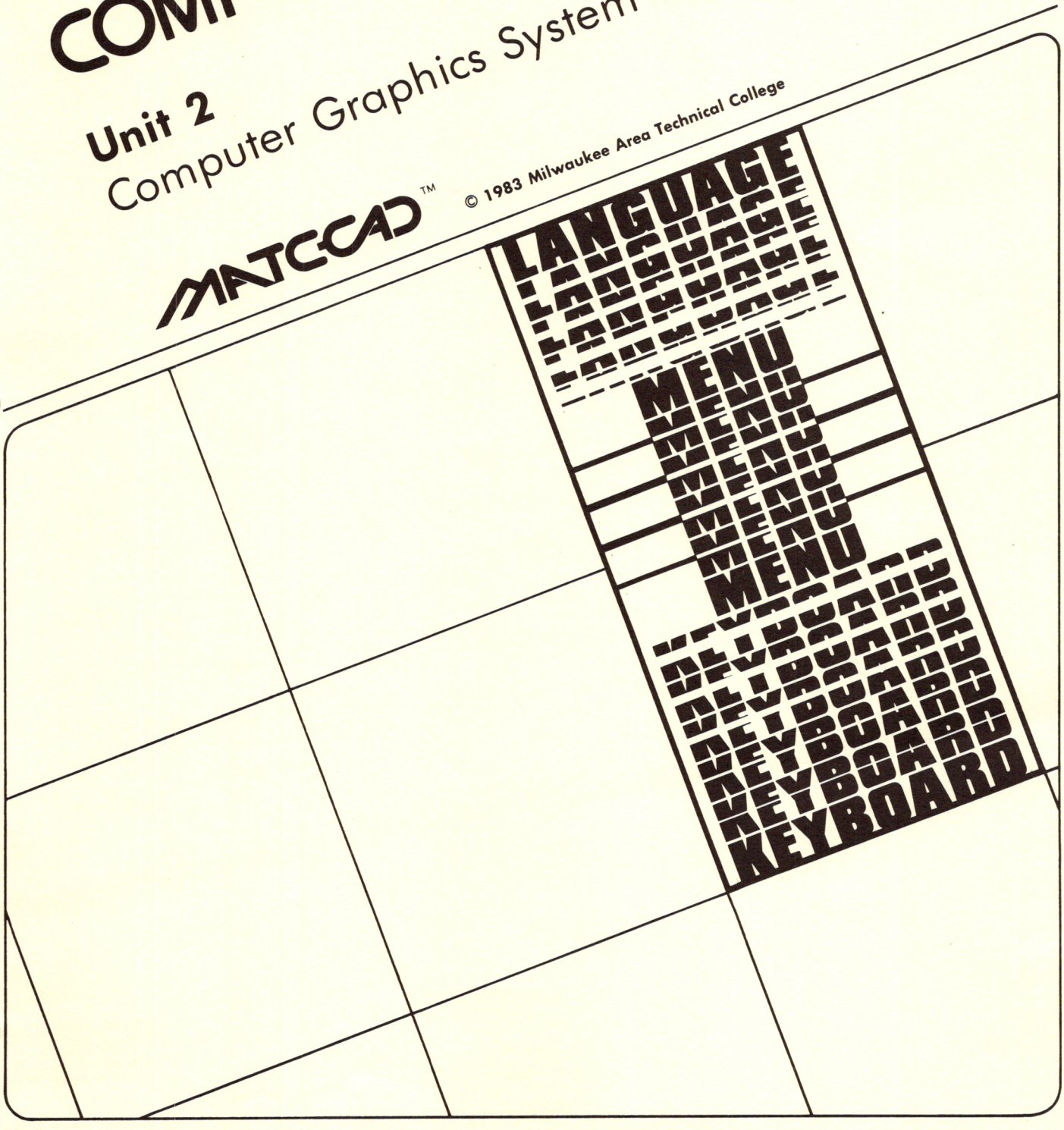

Introduction

This unit will introduce you to the basic operation of the MATC CAD computer graphics design station and how you as an operator can input graphics commands to the computer using the correct command structure. This information is essential to initiating work on the graphics system.

Objectives

At the completion of this unit, you will be able to:

1. Identify the construction (syntax) of a typical graphics command.
2. Identify the functions of the various keyboard symbols and control characters on a computer terminal keyboard.
3. Follow the proper sequence of keyboard commands needed in entering the graphics system, assigning a part name, filing a part for future use and exiting from the graphics system.
4. Given a list of typical graphics terms, define them as they would apply to computer graphics.

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UNIT 2 REFERENCE SECTION

Part I

Command Structure (Syntax) and Vocabulary

The syntax or structure of a typical graphics command is made up of two sections separated by a colon.

Verb-Noun (Modifier) Phrase : Data Input Section

The Verb-Noun phrase tells the computer what you want to do, while the Data Input section indicates the location of the operation to be performed. All graphical commands are entered at the CADDS level with the CADDS prompt (#) displayed.

A. Verb-Noun (Modifier) Phrase

A command always begins with a **Verb** such as Insert, Erase, Move, etc., and, with few exceptions, is followed by a **Noun** such as Line, Point, Circle, etc. **Modifiers**, such as Angle, Length, Tangent to, Vertical, etc., can be added to further specify or restrict a command. Modifiers are always optional and more than one may be used with some commands.

The Verb-Noun (modifier) Phrase is followed by a colon (:).

B. Data Input Section

Data Input refers to the defining of data locations either by digitizing, by coordinate input, or by the selection of a speci fied existing entity.

A data location may be defined in one of three ways:

1. Directly digitizing a location on the graphics tablet with the digitizing pen.
2. Explicitly entering the cartesian or polar coordinates of a desired location.
3. Defining a location based on the position of an existing entity.

C. Executing the Command

The carriage return ( ) is entered at the end of every command. This will execute the command and then return you to the CADDS prompt (#).

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Part II
Symbols, Characters, Punctuation, and Keyboard

A. Symbols

> System level prompt - appears on the display screen when the computer is ready for information at the system level.

\# CADDS system prompt - tells the operator that the CADDS software is activated and that the computer is ready to accept CADDS graphics commands.

⟨CR⟩or ↵ Indicates a carriage RETURN key is necessary at that point.

n Indicates that a numeric value is necessary for the command, with the value to be determined and entered by the user.

↑ In this manual, an upward pointing arrow indicates that the Control key (CTRL) is to be held down while the next key indicated is pressed.

— Underlining: Throughout this manual underlining generally indicates a computer-generated prompt or phrase indicating what mode of input is expected next from the operator.

D_3 Digitizing: Throughout this manual, a subscripted "D" in a command indicates that you are to push the digitizing pen down on the digitizing tablet at some selected or specified location.

← The back arrow key is used to back up one space while entering a part name or inserting text in the CADDS system, or while entering data at the system level.

[ESC] The Escape key is used to switch from a split text-graphics screen, to full screen text, to full screen graphics, and back.

[CAPS LOCK] On the Apple IIe, the CAPS LOCK key must be pushed down as the MATC CAD software will only accept uppercase letters.

B. Control Characters are entered by holding the CTRL key down and then pushing the respective letter key down.

CTRL R - Used to log into the computer at the design station.

CTRL Q - To abort execution of the command that you are currently typing in.

C. Punctuation

The colon (:), semicolon (;), and comma (,) are used for special purposes in the Data Input part of a command. Their function can vary depending on the particular command being used.

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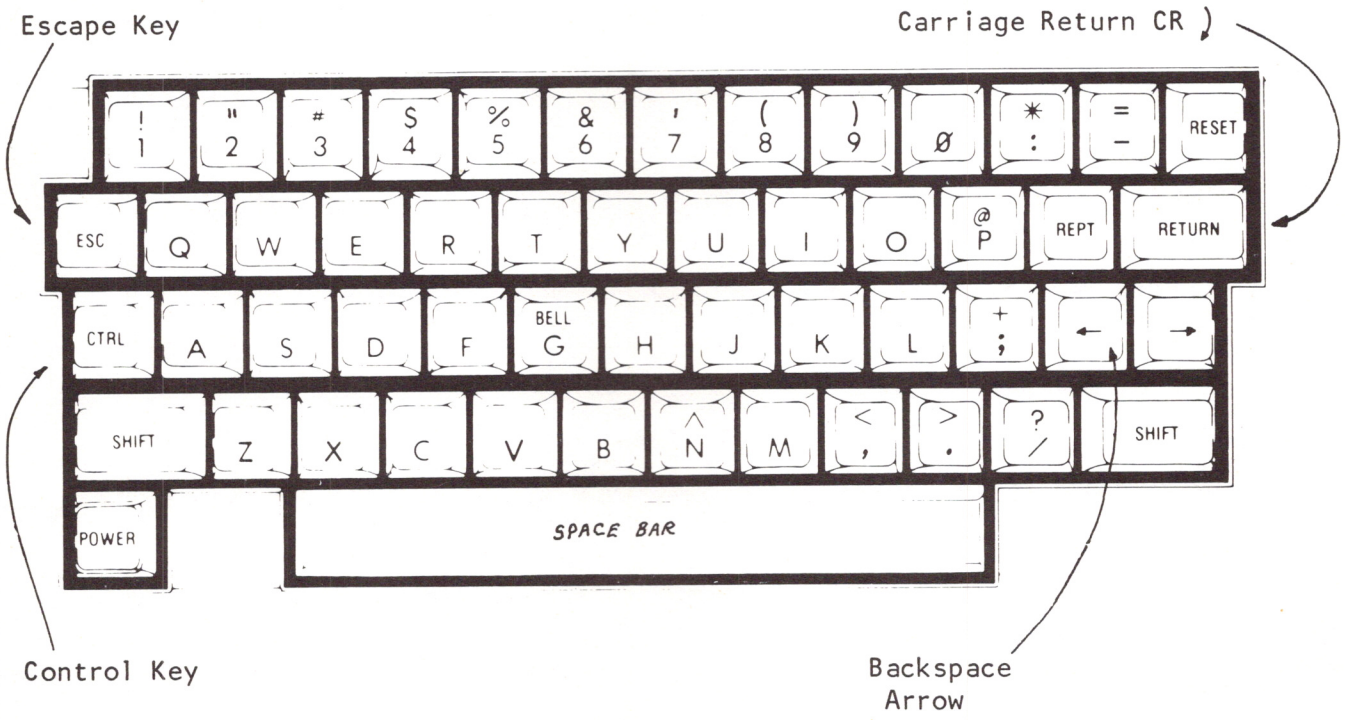
: Colon - When entered at the end of the Verb Noun (Modifier) phrase, this puts the system into the Data Input mode.

; Semicolon - When entered while in the Data Input mode this usually causes any entities already specified to enter the data base. The command then "resets" to the status it was in following the colon and then waits for the new data input from the operator.

, Comma - Commas are typically used to separate one explicitly entered cartesian or polar coordinate from another. When entered to separate digitized input in the Data Input section of a command, the comma sometimes acts exactly like a semicolon (typically when digitizing). When a comma is used as a special limiter in the Data Input section for certain commands, it signals the end of one type of data, then usually prints a DIG prompt and waits for the user to enter the next phase of data input.

D. Keyboard



The Apple II Plus Keyboard

E. Special Commands

RPNT) The repaint (RPNT) command is used to regenerate the display screen image of your entire drawing to clean up the graphics image following entity deletion or modifications.

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This Feature is Present on the Computervision CADDS3® System but has not yet been implemented on MATC CAD.

RECOVER) or # REC) The recover command will enable you to recover the work you have done if the MATC CAD software "crashes" for some reason. When attempting to recover your drawing following a "crash" you must enter the #REC command immediately after logging-in and after indicating that you are working on a NEW part. Any action wuch as calling up an old part, inserting a line, or entering RPNT, will clear out the temporary part area on the disk and you will be unable to recover the drawing.

SEL RPNT OFF)
SEL RPNT ON) Because of the significant time required to regenerate text and dimensions during a repaint, the # SEL RPNT OFF) command has been implemented to allow you the option of blanking out text and dimensions once they have been created. While the # SEL RPNT OFF command is in effect, the text and dimensions will exist in the database but will not appear on the screen.

SEL PRINT ON)
SEL PRINT OFF) The command # SEL PRINT ON will transfer all character input and output so that it will appear on the printer rather than on the screen. This command is useful when verifying and using the on-line documantation. To use this command a printer must be connected to slot #1 in the Apple II.

Spaces in Commands - Spaces are used before the colon in a command to separate the Verb from the Noun from the Modifier. If you are entering a modifier that requires a numeric value such as ANG30, LNG15, PRL2.5, UP1.5, etc., the modifier and associated value must be entered as a continuous sequence of characters without any spaces between.
In the Data Input section of a command after the colon, it is important not to enter any spaces when typing coordinates, masks, etc. In the current version of the software typing a space in the command after the colon may cause problems.

Part Names - When a part name is entered by the user it should consist of from one to seven alphanumeric characters (letters and numbers) and always begin with an alphabetical character. Do not include dashes, spaces, dollar signs, or any other special characters in the part name.

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Part III
MATC CAD Utility Routines

The MATC CAD UTILITY DISK contains routines for the following operations.

1. Formatting blank diskettes for student part storage.
2. Listing the part files on a student disk.
3. Formatting and copying an entire parts disk.
4. Erasing part files from a student disk.
5. Creating a user-defined tablet menu.

Formatting the Part Storage Diskette

The MATC CAD software uses standard Apple II compatible 5 1/4 inch flexible diskettes (single-sided, double density, soft sectored). Prior to first use each part storage diskette must be formatted by the diskette formatting routine. This is accomplished by first inserting the UTILITY DISK into Drive 1 and a blank part storage diskette into Drive 2. After the power is turned on the UTILITY DISK MENU will come up on the display screen giving you the option to choose amoung several operations. To select the FORMAT BLANK DISK option you would enter a 1 (no RETURN key). Formatting a blank diskette takes about 40 seconds. Take care not to touch the disk surface during handling of the disk.

Note: Remember that formatting a disk will erase any drawings or information previously contained on the disk.

Student Disk Part File Listing

To list out the names of the part files that have been created on a student's part disk, the LIST PARTS OR MENU KEYFILES option should be selected from the UTILITY DISK MENU. The computer will then list out the name of each part that has been created, the size of each part in blocks (each block equals 512 characters of information), and the free space remaining on the disk.

Format and Copy the Entire Parts Disk

This option is used to make a copy of the parts disk for backup purposes.

Erasing Part Files From a Student Disk

The ERASE PART option on the UTILITY DISK allows the user to selectively erase one or more parts from the student part disk. After the ERASE PART option is selected, you would then type in the number corresponding to the name of the part to be deleted.

Creating a User-Defined Menu

This option will allow you to create the menu definition file for your own graphics tablet menu. In general there are no restrictions for menu definitions except that the total character count including spaces for a single column of seven menu squares must not exceed 40 characters.

When done with the utility routines, enter the option number to QUIT. Then remove the UTILITY DISK.

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Part IV
Operating the MATC CAD System

The successful operation of any computer-aided drafting system requires that the user be familiar with a wide range of commands allowing the user to create, save, and modify drawings, to move the drawing image around for ease of viewing, and to plot out the final version of the drawing, plus many other functions.

The software features of any computer-aided drafting system fall into distinct conceptual categories based on function. These categories are:

1. Logging into and out of the system, storing and recalling drawings.
2. Geometrical entity creation.
3. Entity modification.
4. Screen image control.
5. Measurement and verification.
6. Text and dimensioning.

Logging Into the MATC CAD Graphics System

Carefully insert MATC CAD Disk 1 and Disk 2 into the disk drives.
Take care not to touch the surface of the disk in the oval cutout area.

Turn Apple computer on. (Switch is at left back corner)

Turn display screen on. (Make sure CAPS LOCK key is down)

Wait 20 seconds for text to appear on screen, and then follow the sequence of commands described below.

SCREEN MESSAGE	USER RESPONSE
WELCOME CAD 1, TO APPLE II PASCAL 1.1	X (no RETURN)
EXECUTE WHAT FILE ?	#5:MATC)
	(Wait 40 seconds)
MATC CAD COMPUTER-AIDED DRAFTING MILWAUKEE AREA TECHNICAL COLLEGE (etc.)	At this point hold down the CTRL key and press R, release both keys, and then press some number 1 to 9.
n>	CADDS)
OLDPART = O, NEWPART = N, REPLY =	N (no RETURN)
PART NAME:	(enter part name, RETURN)
#	(Enter the graphic commands needed to create your part)

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MATC CAD Filing and Log-Out Procedure

When you are finished with your drawing or want to continue working at a later time you will need to file your drawing on your student part diskette using the following procedure.

SCREEN MESSAGE	USER RESPONSE
#	EXIT)
FILE = F, DON'T = D, REPLY =	F (no RETURN)
INSERT YOUR PART DISK IN DRIVE 1 THEN PRESS ANY KEY TO CONTINUE	(insert part disk, press RETURN key)
PART NAME:	(enter part name, press RETURN key)
REPLACE MATC CAD DISK1 IN DRIVE 1, THEN PRESS ANY KEY TO CONTINUE.	(replace DISK 1, press RETURN key)
NEWPART = N, OLDPART = O, GOODBYE = G, REPLY =	G (no RETURN)
n>	LOGOUT)
TASK n TERMINATED	(remove MATC CAD disks, turn off computer, turn off display)

Calling Up an Old Part Drawing for Further Work

Log-in to the MATC CAD system as described on the preceeding page.
After entering the CADDS) command, follow the following procedure.

SCREEN MESSAGE	USER RESPONSE
OLDPART = O, NEWPART = N, REPLY =	O (no RETURN)
INSERT YOUR PART DISK IN DRIVE 1 THEN PRESS ANY KEY TO CONTINUE	(insert part disk, press RETURN key)
PART NAME:	(enter your old part name, then press the RETURN key)
ENTERING OLD PART DATA BASE UNIT IS:IN	
REPLACE MATC CAD DISK1 IN DRIVE 1, THEN PRESS ANY KEY TO CONTINUE.	(replace DISK 1, press RETURN key)
#	(Continue work on your drawing)

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Filing Your Drawing and Resuming Work

It is a good idea to file the drawing you are working on every 20 to 30 minutes. This way if something happens you will have only lost a few minutes rather than a whole day's work. This "insurance filing" of your drawing is done by entering the # FILE ↵ command. The computer will ask you for the name under which you want to file your drawing, prompt you to insert your student part disk, and then proceed to save a copy of the present form of the drawing. If you are especially concerned about losing the work you have done so far on a drawing, you can either use two diskettes and save your work on alternate disks, or you can alternate the names under which you save the drawings.

Filing an Updated Version of Your Part Drawing

SCREEN MESSAGE	USER RESPONSE
#	EXIT ↵
FILE=F; DON'T=D; REPLY=	F (no RETURN after F.)
FILING NAME:	A RETURN at this point will use the old part name.
WED.EX1	
PART ALREADY EXISTS TYPE OK TO FILE ANYWAY	
	Since your part was previously existing on the disk and you would like to save the updated version you will type OK ↵
NEWPART = N; OLDPART = O; GOODBYE = G; REPLY =	G (no RETURN after G)
3>	LOGOUT ↵
TASK 3 TERMINATED	

Part V
Crash Recovery

This Feature is Present on the Computervision CADDS3® System but has not yet been implemented on MATC CAD.

MATC CAD system allows the user to recover his/her part after a system "crash" due to software problems or power failure. If a system crash occurs or the user shuts off the computer power before a part is filed, the work done up to that time is still present in a scratch file area on DISK 2 . When the user starts up the MATC CAD software later and enters CADDSCLR or CADDS, the work done prior to the crash can be recovered by entering the command # RECOVER ↵. If the part is recoverable,the part image will then appear on the screen and the user can continue working. When finished, the user can EXIT and file the part in the normal manner, making sure to type in the name under which the part is to be filed.

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Part VI
Computer Graphics Terms and Definitions

TERMS	DEFINITIONS
CAD/CAM	Computer Aided Design/Computer Aided Manufacturing
CADDS3	Name of Computervison Corp. CADDS3 software package. "Computer-Aided Design and Drafting System 3"
COMMON FILE (COM FILE)	A file made of the mathematical descriptions of all entities that make up a part. (Your part is stored in this form on the disk.)
CPU	Central Processor Unit, the main part of a computer.
CRT	Cathode Ray Tube (the display screen).
CURSOR	A movable indicator symbol on the graphics display that shows the current position of the digitizing pen.
DATA BASE	A generalized way of representing and storing a drawing on the magnetic disc which allows access of the information contained in your drawing for multi-purpose applications such as in manufacturing, accounting, etc.
DATA INPUT	The Data Input section follows the colon in a command. In this mode you locate a position on the display screen through digitizing or explicit coordinate input.
DEFAULT VALUE	The value of parameters such as screen scale, text height, etc., to which the graphics system is initially set by the computer system when starting a new part.
DIGITIZING	The operation of selecting a location on the graphics display by pushing the electronic digitizing pen down on the digitizing tablet at a user-selected position.
DIGITIZING TABLET	Device to enter commands and data locations by means of the digitizing pen.
ENTITY	A general term for an individual line, circle, fillet, point, etc.
GRAPHICS DISPLAY	A CRT (Cathode Ray Tube) unit that displays the graphic entities in your drawing.
INTERACTIVE	A type of user-computer interaction where a request or command from the user is immediately recognized and acted upon by the computer.

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KEYBOARD	A computer terminal unit used to enter commands and data.
MASK	A "modifier" that is used in the Data Input section of a command to limit the selection of an entity to the particular type of entity desired (ex: LIN, CIR, PNT).
MNEMONIC (ne mon'ik)	A three- or four-letter abbreviation that represents a word in the graphics command vocabulary.
MODIFIER	Used in a graphics command following the Verb and Noun to augment, further specify, or limit what is to be accomplished or selected by the Verb-Noun combination.
PARAMETERS	Values of certain system features such as text height, grid size, line font, screen scale, etc. (Parameters are usually set to some value and are saved along with your part.)
PROMPT	A three letter abbreviation (DIG, ENT) generated by the computer in the Data Input section of a command that lets you know what type of information is expected next from the user.
REGEN FILE	A file used by the Computervision graphics system to represent the "Picture image" of a part. (Not used with the MATC CAD system.)
SOFTWARE	A collection of computer programs that operates the computer and allows the user to do mechanical, electrical, etc., applications.
SYNTAX	The structure of a graphics language command.

Part VII
Using The Graphics Tablet Menu

Most computer graphics systems have a facility called a **menu** that allows the user to select commands or sections of a command by pushing the digitizing pen down in the menu area of the digitizing tablet. With the MATC CAD system, the menu area occupies the upper third of the digitizing tablet and consists of 154 one-half inch squares arranged in 7 rows by 22 columns. Other graphics systems have different sized menu areas.

Different menus can be created depending on the application, and for this course a training menu has been provided.

To call up and activate the menu you must type in the command: # RST KEY) .
The system should then respond with the message: ALL KEYFILES RESTORED.
A copy of the menu supplied with MATC CAD is shown in Figure 1.

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VERB			NOUN			MODIFIER				MASK												
INS	MOV		LIN	PNT		INTOF	TANTO	PPNT	.	END	DIG		X	IX	7	8	9	FLIP SCREEN		RETURN	
ERS	ROT		CIR	ARC		PRL	PRP	R	.	PNT	LIN	ALL	Y	IY	4	5	6			REPAINT	
TRIM	COPY		FIL	TXT		LNG	ANG	D	,	ENT	CIR	WIN	R	IR	1	2	3	SEL ON	RPNT OFF		CTRL Q
DIV	MIR		LAY	ALL		SUP	TOL		;	ORG	TXT		A	IA	—	.	0				
SEL	CHG		DIM	GRD		HOR	VER	ON		REF				N		SEL GRD G	SEL GRD ORG	PAG DIS	ZOM WIN		
VER	ECHO		LDIM	ADIM		HGT	PREC	OFF		SEL PRINT	PRINT		FILE	EXIT	SAV DIS	ECH GRD ON	ECH GRD OFF	ZOM UP	ZOM ORG		
LIST			RDIM	DDIM		AO	AI		LAY USED	LIST LAYER	GRD	PARAM	MEASURE DIS	ANG	RST DIS	SEL GRD ON	SEL GRD OFF	ZOM DWN	ZOM ALL		

Figure 1. Menu supplied with MATC CAD

Part VIII
Printer Plotting

If the Apple computer you are using is connected to an Epson dot matrix printer, you can print out an image of what is visible on the graphics display screen.

The command to do this is:

PLOT)

Each dot on the screen will be printed out as one dot on the printer giving an image of about 3" x 5" with 280 dots horizontally by 192 dots vertically. Because the horizontal and vertical spacing of the dots is not equal on the printer, the printed image will be compressed slightly vertically giving ellipse-shaped circles.

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UNIT 2 ASSIGNMENT SECTION

Part I
Punctuation and Keyboard Exercises

Directions: Give the proper symbols for the following:

SYMBOL

A. system level prompt

B. carriage return

C. control key

D. representation of a number

E. CADDS level prompt

Directions: Give the proper symbol or command for the following:

A. To abort a command.

B. To log into the Operating system.

C. To backspace in the Operting system (system prompt " ") or when inserting text in the CADDS system.

Directions: Define the functions of the following punctuation characters in the CADDS System:

: (colon) -

; (semicolon) -

, (comma) -

Part II
Log In and Log Out Exercise

Directions: Fill in the blanks for the following sequences of operations for logging into the system.

A._____ R _____

B. n> _____)

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C. NEW PART = N, OLDPART = O, REPLY = _____

D. PART NAME: _____)

Directions: Fill in the blanks for the following sequence of operations for filing a part and logging out of the system.

A. _____)

B. FILE = F, DONT = D, REPLY = _____

C. FILING NAME: _____

D. NEW PART = N, OLDPART = O, GOODBYE = G, REPLY = _____

E. 2> _____

Part III
Command Syntax Definitions Exercise

Directions: Define the following in terms of their function and how they are used in the graphics command language (syntax).

A. System prompt

B. Verb-Noun Phrase

C. Colon

D. Data Input Section

E. Execution of a Command

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Part IV
Terms and Definitions Exercise

Definition	Term
a. A computer terminal unit used to enter commands.	_____
b. Three- or four-letter abbreviation that represents a word in a graphics language vocabulary	_____
c. Structure of commands in a graphics language.	_____
d. A type of user-computer interaction where a request by the user is acted upon immediately by the computer.	_____
e. The indicator symbols on the graphic display that shows your position.	_____
f. Cathode Ray Tube	_____
g. Computer Aided Design/Computer Aided Manufacturing	_____
h. Selecting locations on the screen using the electronic pen on the digitizing tablet.	_____
i. General term for a line, circle, fillet, point, etc.	_____
j. A Data Input section modifier used to screen out unwanted entities.	_____
k. Device to enter commands and data by means of the digitizing pen.	_____
l. Central Processor Unit	_____
m. Values of certain system functions: text height, form, line font, fillet size, etc.	_____
n. The system-selected value of a parameter such as screen scale, etc.	_____
o. CRT unit that displays specified graphic entities.	_____
p. A collection of computer programs that operate the computer and allow the user to do mechanical, electrical, etc., applications.	_____

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UNIT 3 ASSIGNMENT SECTION

Part I
Line Exercises

Directions: Write the language as it would appear on the command display and specify the location of the digitizes that would be needed to construct the following:

Example:

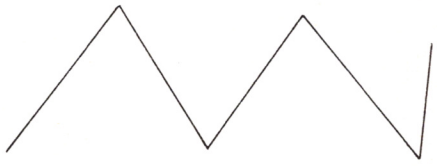
Command Language

INS LIN : DIG D_1 D_2)

1.

1.

2.



2.

3.

3.

4. (This line is horizontal)

4.

5. (This line is vertical)

5.

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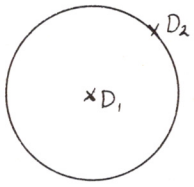
Part II
Circle Exercises

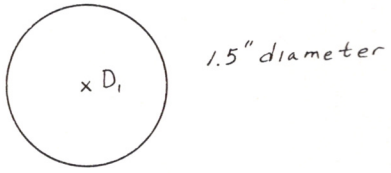
Directions: Write the language as it would appear on the command display and indicate the location of the digitizes that would be needed to construct the following:

1.



1.

2.



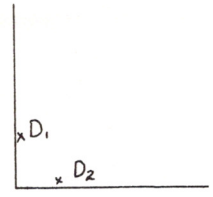
2.

Part III
Fillet Exercises

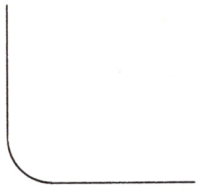
Directions: Write the language as it would appear on the command display and indicate the location of the digitizes that would be needed to construct the following:

Example:

Before



After



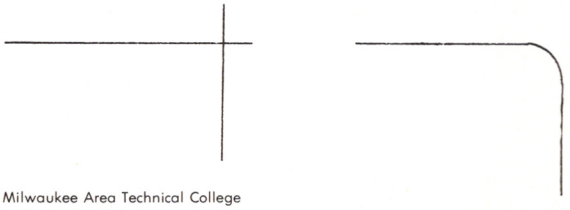
Command Language

```
# INS FIL : ENT D1   D2   )
# RPNT )
```

1.

1.

2.



2.

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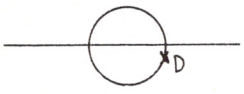
Part IV
Erasing Exercises

Directions: Write the language as it would appear on the command display and indicate the location of the digitizes that would be needed to erase the following:

Example:

Before

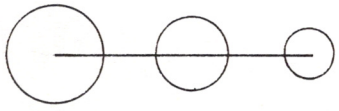


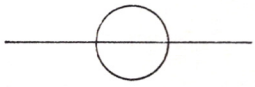
After

Command Language

ERS CIR : ENT D)
RPNT)

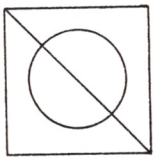
1.





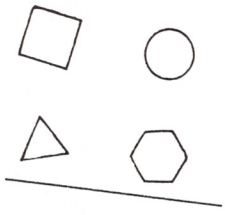
1.

2.



2.

3.



blank screen

3.

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UNIT 2 LAB EXERCISES

Lab Exercise A

This exercise will introduce you to the proper sequence of commands needed to create and save a drawing on the MATC CAD graphics system. For this exercise you will go through the sequence of:

1. Logging into the MATC CAD system and naming a new part.
2. Creating a drawing.
3. Filing the drawing on your part disk and logging out of the system.

Carefully insert MATC CAD Disk 1 and Disk 2 into the disk drives.
Take care not to touch the surface of the disk in the oval cutout area.

Turn Apple computer on. (Switch is at left back corner)
Turn display screen on. (Make sure CAPS LOCK key is down)

Wait 20 seconds for text to appear on screen, and then follow the sequence of commands described below.

SCREEN MESSAGE	USER RESPONSE
WELCOME CAD 1, TO APPLE II PASCAL 1.1	X (no RETURN)
EXECUTE WHAT FILE ?	#5:MATC) (Wait 40 seconds)
MATC CAD COMPUTER-AIDED DRAFTING MILWAUKEE AREA TECHNICAL COLLEGE (etc.)	At this point hold down the CTRL key and press R, release both keys, and then press some number 1 to 9.
n>	CADDS)
OLDPART = O, NEWPART = N, REPLY =	N (no RETURN)

PART NAME: At this time enter a part name from one to seven characters in length that your instructor has told you to use. Do not make up just any name and remember not to include any spaces, dashes, or special characters in the name.

CREATING NEW PART
#

You will be creating the following drawing with all the needed commands given. Make sure that you type in the commands exactly as shown, paying attention to colons, commas, and any spaces in the command ahead of the colon. Remember that D D , etc., in a command means that you push down the digitizing pen on the digitizing tablet where indicated in the drawing. The D D , etc., are not typed in on the keyboard. Pay special attention to the differences between the letters I and O and the numbers 1 and 0. The underlined words DIG or ENT will be printed out by the computer and should not be typed in.

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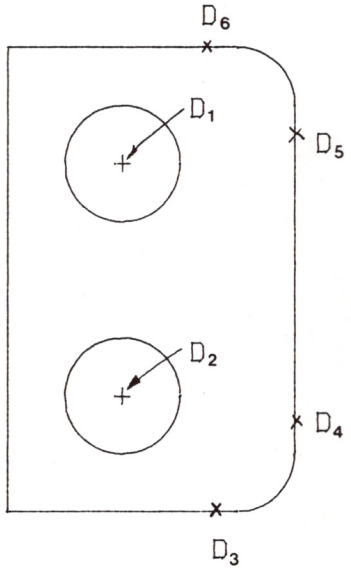
Note: D D D etc., in the following commands indicate that you are to press down with the digitizing pen on the digitizing tablet at the location indicated in the drawing.



Graphic Commands	Comments
# INS PNT : DIG X1Y1, Y3)	This will put in 2 points.
# ZOOM ALL P200)	Center the points on the screen.
# INS LIN : DIG X0Y0, X2.5, Y4, X0, Y0)	This will create the four lines of the rectangle.
# ZOOM ALL P100)	
# INS CIR R.5 : DIG PNT D_1 DIG PNT D_2)	This will put in circles centered at existing points. Make sure that you type in PNT after each DIG prompt.
# INS FIL : ENT D_3 D_4 D_5 D_6)	Be especially careful to digitize the sides in the order shown.
# RPNT)	This will redraw the part image.

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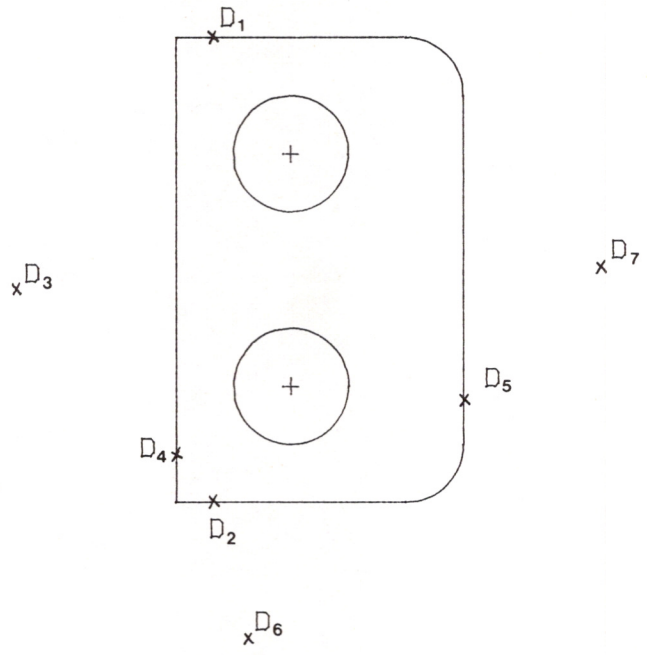
For the next part of the exercise you will be adding some dimensioning and text to your drawing.

All digitizing should refer to the second illustration.



Graphic Command	Comments
# ZOOM ALL P100)	
# INS LDIM VER : ENT D_1 D_2 DIG D_3)	Vertical dimension.
# INS LDIM HOR : ENT D_4 D_5 DIG D_6)	Horizontal dimension.
# INS TXT /your name/ : DIG D_7)	Enter your name between slashes.
# RPNT)	Repaint to clean up drawing.
	(If everything looks correct at this time, then type in EXIT.)
# EXIT)	
FILE = F; DON'T = D; REPLY = D	D No RETURN after D
NEWPART=N; OLDPART=O; GOODBYE=G; REPLY =	G No RETURN after G
3> LOGOUT)	
TASK 3 TERMINATED	

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COMPUTER-AIDED DRAFTING

Unit 3

Basic Entity Creation—Lines,
Circles, Points, Fillets, Text, and Dimensions

MATCCAD™

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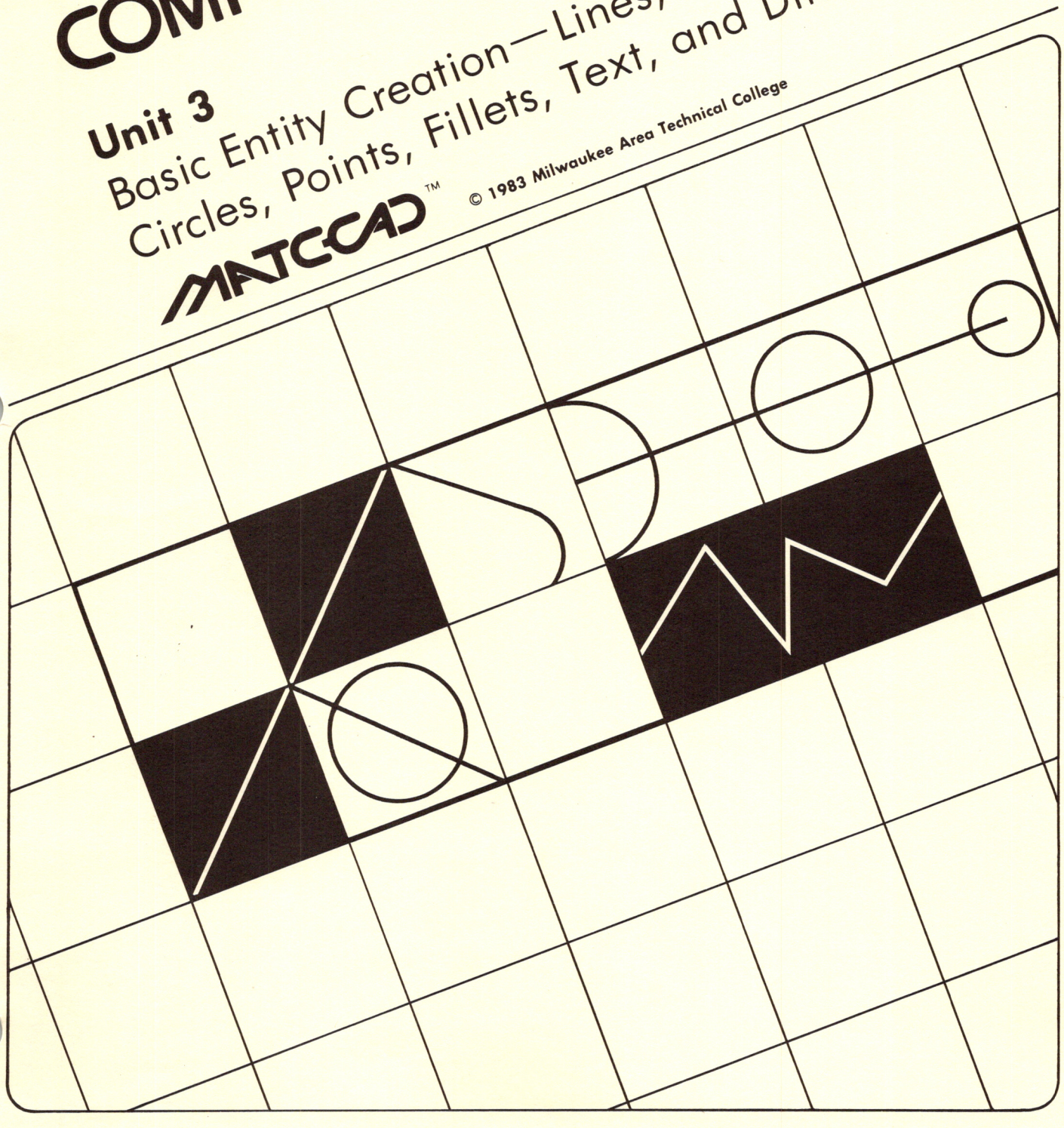

Introduction

This unit will introduce you to the basic methods for creating lines, circles, points, fillets, text, dimensions, and splines. This will give you a foundation of technique and experience to build upon as you progress toward the more difficult exercises and applications that will follow.

Objectives

At the completion of this unit you will be able:

1. To give the graphics commands needed to create basic lines, circles, points, fillets, text, and dimensions using the graphics system.

2. When given the commands that would create various lines, circles, fillets, and points, to draw these entities as they would appear on the data display screen of a graphics design station.

3. When given different entities; to write the command language and show the digitizes that would be needed to erase them.

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UNIT 3 REFERENCE SECTION

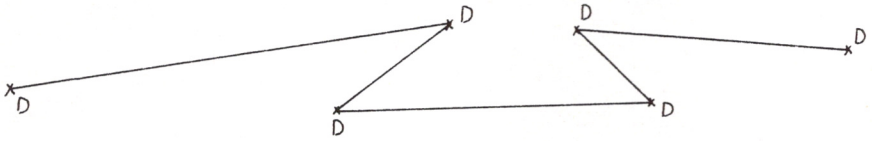
Part I
Basic Line Insertion

The Insert Line command connects two specified locations. Each line has two ends. The origin of a line is at its midpoint.

A. Basic Line Command

Prompt	Verb	Noun	Mod	Colon	Mode Prompt	Execute
#	INS	LIN		:	DIG D_1 D_2	)

For an unbroken sequence of lines continue to digitize. D D D D D D D)



To move to a new position to start a new line, enter a semicolon to separate the lines. D_1 D_2 ; D_3 D_4)

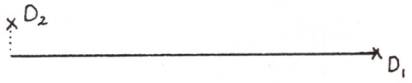
B. Basic Line Modifiers

VER Forces the line to be inserted vertically.
HOR Forces the line to be inserted horizontally.

COMMAND

Prompt	Verb	Noun	Mod	Colon	Prompt	Execute
#	INS	LIN	VER	:	DIG D_1 D_2	)

#	INS	LIN	HOR	:	DIG D_1 D_2	)



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Part II
Basic Circles

The Insert Circle command can insert circles in five different ways. A circle is generated in a counterclockwise rotation with its beginning and end at zero degrees (3 o'clock). A circle's origin is at its center.

A. Basic Circle Command

Prompt	Verb	Noun	Modifiers	Colon	Mode Prompt	Digitize	Execute
#	INS	CIR		:	DIG	D D D	)

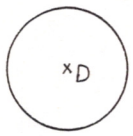
B. Basic Circle Modifiers

The two modifiers that are used most often with the Insert Circle command are:

1. R followed by a value "n" representing an explicit radius
2. R by itself, indicating that the radius is specified by the second digitize

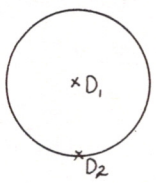
When the actual radius is specified following R, only one digitize is needed which indicates the center of the circle.

INS CIR R2.5 : DIG D)



When the modifier R is used by itself without a specified radius, two digitizes are needed. The first digitize locates the center, and the second digitize will indicate the radius of the circle.

INS CIR R : DIG D_1 D_2)



Part III
Inserting Points

A point is a non-dimensional entity occupying a single X Y coordinate. It may be inserted anywhere in the part and used as a reference or insertion point for other entities. It is indicated on the graphic display as a plus sign (+) of constant screen size.

A. Basic Point Command

Prompt	Verb	Noun	Modifier	Colon	Mode Prompt		Execute	
#	INS	PNT		:	DIG	D_1	)	$+D_1$
For multiple points:					DIG	D D D D	)	

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Part IV
Basic Fillets

A fillet is an arc inserted at the intersection of two entities usually to round off an inside or outside corner. These entities may be lines, points, arcs, or circles. If the two entities do not intersect, a fillet will be inserted at their projected intersection. When inserting a fillet between lines, the lines will be automatically trimmed shorter or longer to end at the fillet.

Important: A fillet, like a circle or an arc, is formed in the counterclockwise direction; therefore, in digitizing entities to form a fillet the digitizing must be so that the fillet will be formed counterclockwise. Take care to select the two lines as close as possible to the expected fillet tangency locations.

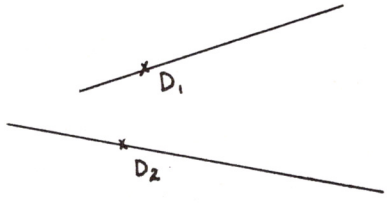
A. Basic Command

INS FIL Rn : ENT D_1 D_2)

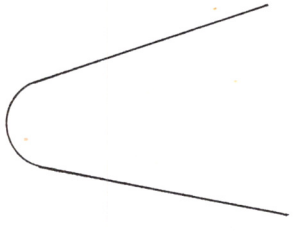
where "n" is a radius specified for the fillet
(a .5 radius is used as the default if "Rn" is omitted)

INS FIL R.5 : ENT D_1 D_2)

Note that entity selection is done in a counterclockwise direction.



Before



After

Part V
Erase Command

The erase command is used for removing entities from a drawing. Entities may be erased by identifying each one, by erasing all of a certain type, or by erasing the entire screen.

A. Specific Entities

ERS (entity type) : ENT D D D D)
Will erase only digitized entities of the specified type.

EXAMPLES

ERS LIN : ENT D)
ERS PNT : ENT D)
ERS CIR : ENT D)
ERS TXT : ENT D)

ERS ALL: ENT D D D D)
Will erase any entity selected by digitizing.

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B. All Entities

ERS ALL) (Computer Printout) ARE YOU SURE? (You enter: Y for yes)

All entities will be erased.

Note: Any time you get the response: "ARE YOU SURE?" the computer thinks that you want to erase everything that exists. Press N if you don't.

Part VI
Modes and Masks

A. Data Input Mode Prompt

After entering a colon following the verb-noun part of a command the computer will respond with either DIG or ENT mode prompt indicating what mode of input is expected from the operator at that point in the command.

DIG - A DIG prompt indicates that you are to indicate some position on the graphics display either by pushing the digitizing pen down on the tablet at some location or entering some X Y coordinate value.

ENT - An ENT prompt indicates that the computer expects you to select an existing entity such as a line, circle, or point.

You as an operator can change the mode of Data Input in a command when appropriate by typing in a different mode mask. In the following example the END mode mask is used to insert a line that begins at the end of the existing line. Note that the DIG mask must be entered by the user prior to IX3. Otherwise, the computer will look for the end of another line three inches over from the end of the first line, which it will not find.

INS LIN : DIG END D_1 DIG IX3)

Before After

Note: It is important to remember that when you insert or change a mode or entity mask in a command, that particular mask will remain in effect until you change it again or the computer switches the mask.

B. Masks

A mask is entered by the operator to limit selection by the computer to entities of a particular type, or to a particular area of the screen, or to a position on the end, or in the center of existing geometry.

Masks come in three varieties:

1) entity selection 2) positioning 3) area masking.

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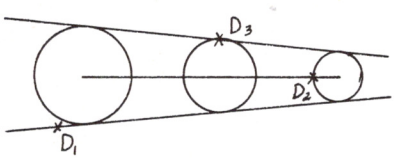
1. Entity selection masks - LIN, CIR, PNT.

Entity selection masks are used to limit the selection of entities to only those entities of the particular type specified by the mask.

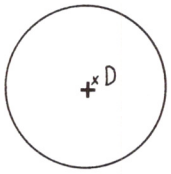
ERS ALL : ENT LIN D_1 D_2 CIR D_3)



Note that even though the circle and lines were very close to the first two digitizes, only the lines were removed because of the LIN mask. Digitize 3 is on a line but because of the CIR mask only the circle is erased.

INS CIR R1: DIG PNT D)

In this example the circle is inserted so that its center is at an existing point.



2. Positioning Masks - END, ORG.

Positioning masks are used so that the operator can reference entities based on the location of existing entities.

END and ORG Masks

When the **END** mask is inserted in the Data Input part of a command, it enables the operator to select the end point of an existing entity by digitizing anywhere near the half of the entity closest to the end desired.

INS LIN : DIG END D_1 D_2)

When the **ORG** mask is inserted in the Data Input mode it enables the operator to select the origin of an existing entity by digitizing anywhere near that entity.

The origin of a line is at the center of that line. The origin of a circle is its center.

INS LIN : DIG ORG D_1 D_2)

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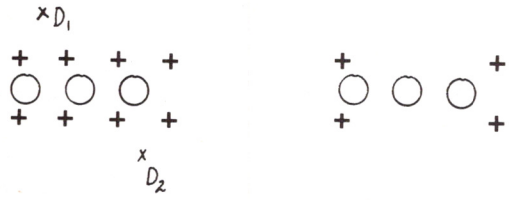
3. Area Masking - WIN

Windowing can be used as a mask in a command to select all entities (perhaps of a particular type) that are within an area. This can be of great help when doing some large scale operation such as erasing an area, changing the size of a group of dimensions or moving a group of entities.

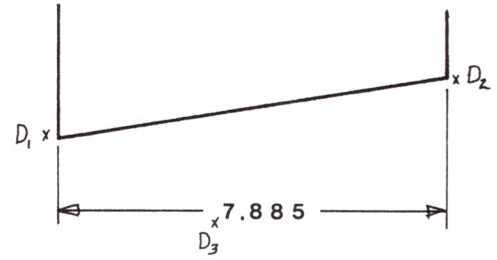
ERS PNT : ENT WIN D_1 D_2)



Part VII
Basic Dimensioning

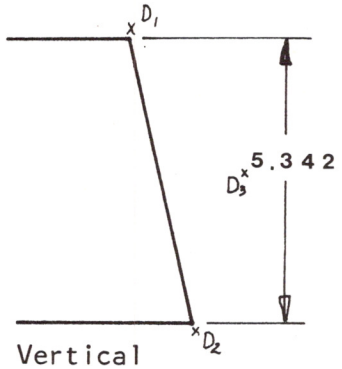
You can automatically dimension the horizontal or vertical distance from the end of one entity to another. Two digitizes are required to select the two reference ends of the dimension measurement and a third digitize is then used to locate the text of the value of the dimension. The dimension lines, extension lines, arrowheads, and dimension text are all automatically inserted at an appropriate size. The default precision for the dimension is 3 decimal digits.

INS LDIM HOR : ENT D_1 D_2 DIG D_3)



Horizontal

INS LDIM VER : ENT D_1 D_2 DIG D_3)



Vertical

Part VIII
Basic Text Insertion

The Insert Text command is used to specify text and its location.

1. Basic Command:

INS TXT /TEXT GOES HERE/ : DIG D)

space required after TXT

Delimiters - each string of text is delimited at the beginning and end by a single quote, double quote or slash (' " /).

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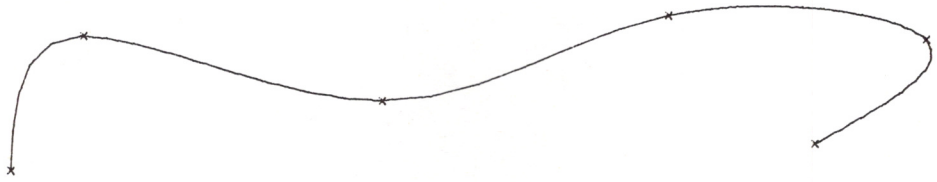
Part IX
Splines

This Feature is Present on the Computervision CADDS3® System but has not yet been implemented on MATC CAD.

A spline can be used to fit a smooth curve through a series of points. Splines can be used for graphs, illustrations, or anywhere that smoothly irregular-shaped curved line is needed.

INS SPL : DIG D D D D D)



Part X
Basic Screen Control

The screen control commands are used to get the screen image of your drawing on the display to a scale and position convenient for you to work on. Some of the basic screen control commands are:

#ZOOM ALL)

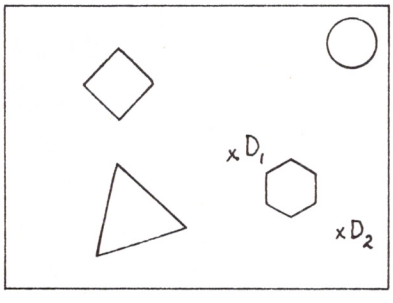
This will fill the screen edge to edge with your drawing.

#ZOOM ALL P30)

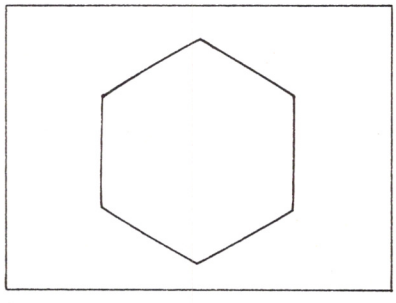
This will put your part on the screen with a 30% blank border on all sides.

#ZOOM WIN: DIG D_1 D_2)

Two opposite corners of a rectangular area are digitized on the graphics tablet. The computer will enlarge the rectangular area to fill the screen.



Before



After

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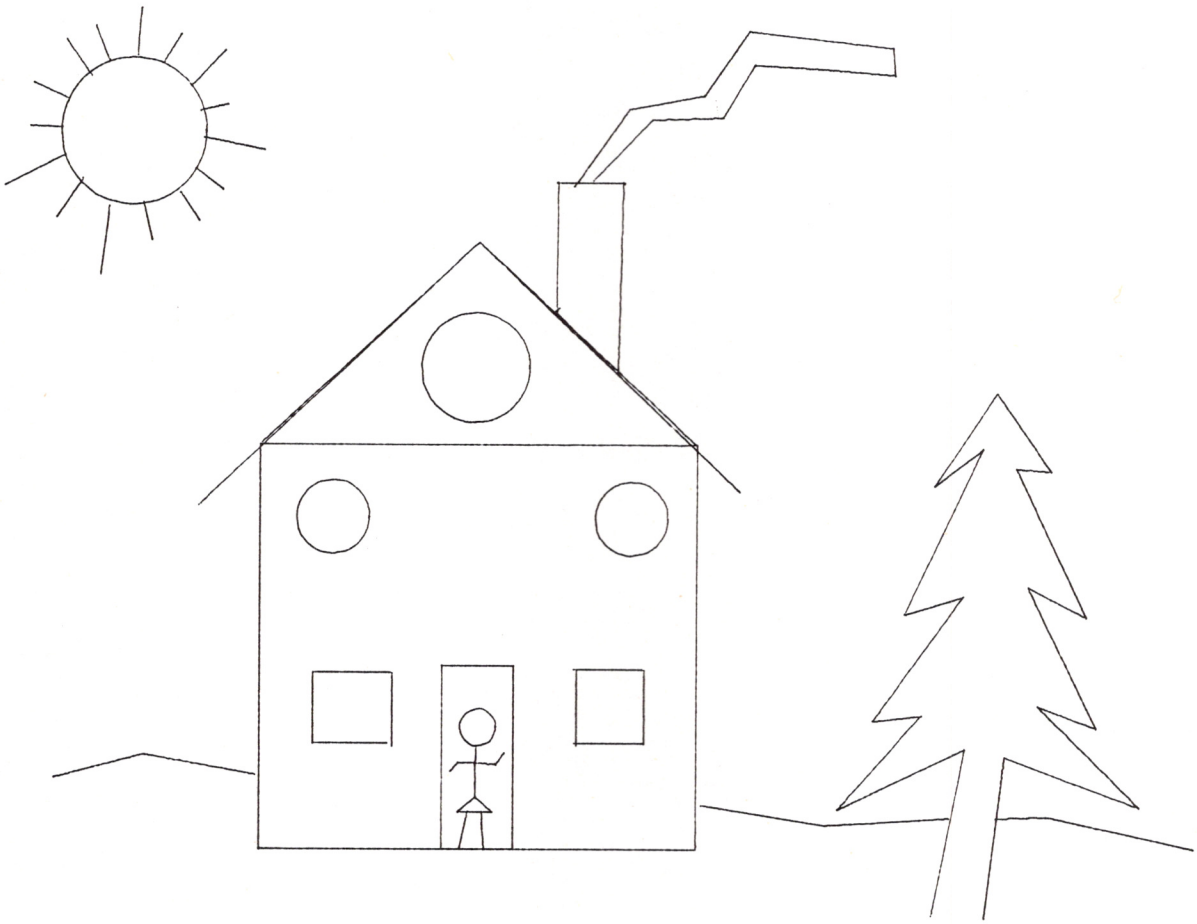
UNIT 3 LAB EXERCISE

Directions: Construct the following drawing on the data display. Use the part name _____. After construction is completed, file the part, log-out, recall the part from the file and have your instructor check it.



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COMPUTER-AIDED DRAFTING

Unit 4
Using Screen Control Commands (Zoom, Page, Save, and Restore Display)

MATCCAD™

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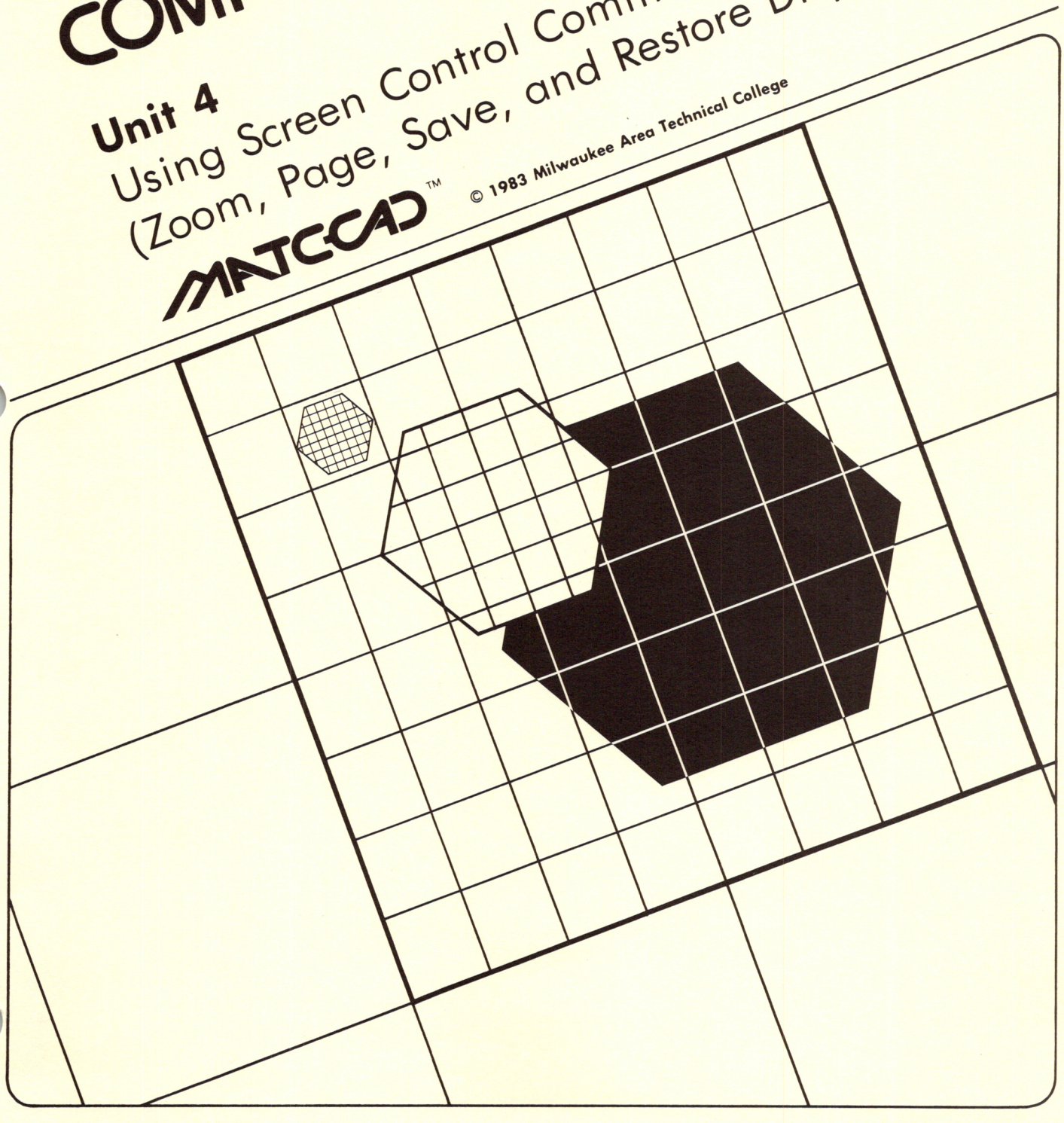

Introduction

The screen control commands zoom, page, save display, and restore display are used to view and manipulate the position and scale of the displayed image of your drawing on the graphics display screen. With these commands you can enlarge, condense, shift over the image, or save a screen image at a particular scale and position for later recall.

Objectives

At the completion of this unit you will be able:

1. To use the screen control commands ZOOM and PAGE to position and scale the displayed image of your drawing to a convenient area and size for you to work on.

2. To use the Save Display and Restore Display commands to save and later recall a previously displayed image of your drawing at a particular screen position and screen scale.

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Unit 4 Reference Section

Part I
Zoom Commands

The ZOOM commands are used to view a part at different screen scales. The screen scale determines how large the drawing appears on the graphics display in relation to the actual dimensions of the drawing. The actual dimensions of the part remain intact regardless of what scale it is displayed at on the screen.

ZOM ALL ↵ — Your drawing will be rescaled to fill the screen edge to edge or top to bottom.

ZOM ALL Pn ↵ — Your drawing will be scaled so that there is a blank border on each side of the drawing equal to some percentage "n" of the total width (or height) of the actual drawing. P100 will give a border on each edge equal to the width or height of the part. P30 will give a border equal to 30% of the width or height of your part.

ZOM UP ↵ — Doubles the image size of your drawing on the screen.

ZOM DWN ↵ — Shrinks the image size of your drawing to one half of its prior size.

ZOM UP Rn ↵ — Enlarges the screen image by a ratio of n (n = any positive number). Example: ZOM UP R5 ↵ will display the drawing image five times as large.

ZOM DWN Rn ↵ — Shrinks the screen image by a ratio of "n" to one.

ZOM WIN : DIG D D ↵ — D and D represent two opposite corners of a rectangular window area which will then enlarged to fill the screen.

ZOM ORG ↵ — Will redraw the part image to the original screen scale with X=0 Y=0 at the center of the screen.

ZOM ALL SCLn ↵ — Will set the displayed scale of the part so that the plotted image will come out at the desired scale. SCL1 will give a full-sized plot. SCL.5 will give a half-sized plot. The plotter to be used must be selected before the ZOM ALL SCL command. The #ZOM SCLn command will only work on the plotter disk.

This Feature is Present on the Computervision CADDS3® System but has not yet been implemented on MATC CAD.

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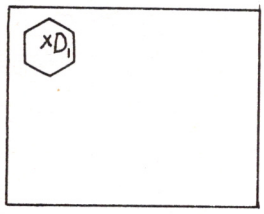
Part II
Page Commands

This command repositions the image of your drawing on the display screen while retaining the original XY coordinates of each entity in the drawing.

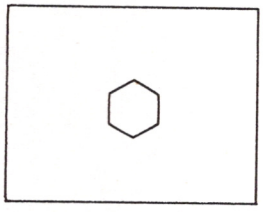
PAG DIS : DIG D_1)

When one location is digitized following the DIG prompt, the location is shifted to the center of the screen.

Before



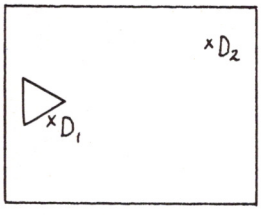
After



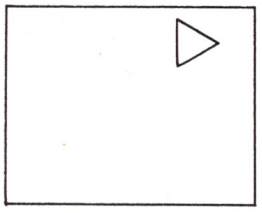
PAG DIS : DIG D_1 D_2)

When two locations are digitized following the DIG prompt, the first location digitized is shifted to the second location digitized.

Before



After



Part III
Saving and Recalling Screen Images

The Save Display command is used to save the current displayed screen image in terms of the current screen scale and position of XOYO in your drawing. Each display that is saved is given some number from 1 to 4.

Example: # SAV DIS 3)

At any later time you can recall a previously saved view by typing in
RST DIS n)
where "n" is the number of the display you previously saved.

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UNIT 4 EXERCISE SECTION

Part I
Zoom and Page Exercises

Directions: Write the command as it would appear on the command display in order to do the following exercises:

	Command Language
Example: To fill the graphic display screen with everything that has been worked on.	# ZOM ALL)
1. To increase the displayed size of the image on the graphic display by a factor of four.	1._____
2. To decrease the displayed size of the image on the graphic display by a factor of ten.	2._____
3. To double the displayed size of the image on graphic display.	3._____
4. To set the image size on the screen at a life size scale.	4._____
5. To halve the displayed size of the image on graphic display.	5._____
6. To fill the screen with the image of a part bounded by X1Y1 and X10Y8. (XY coordinates are covered in Unit 6.)	6._____
7. To reposition X7Y5 to the exact center of the screen.	7._____

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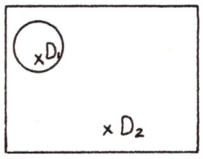
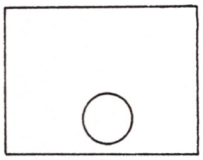
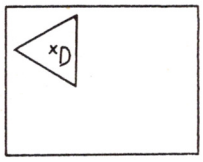
Directions: Write the command language as it would appear on the command display and indicate the digitizes needed for the following exercises:

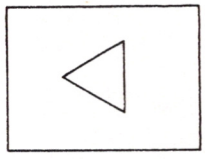
Example: Command Language

Before 

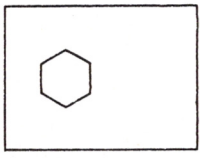
After 

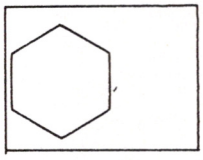
PAG DIS : DIG D_1 D_2)

1. 

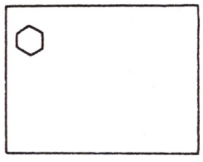


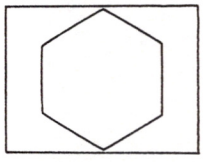
1. Move to center of screen

2. 



2.

3. 



3.

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UNIT 4 LAB EXERCISE

Lab Exercise 4

Objectives:

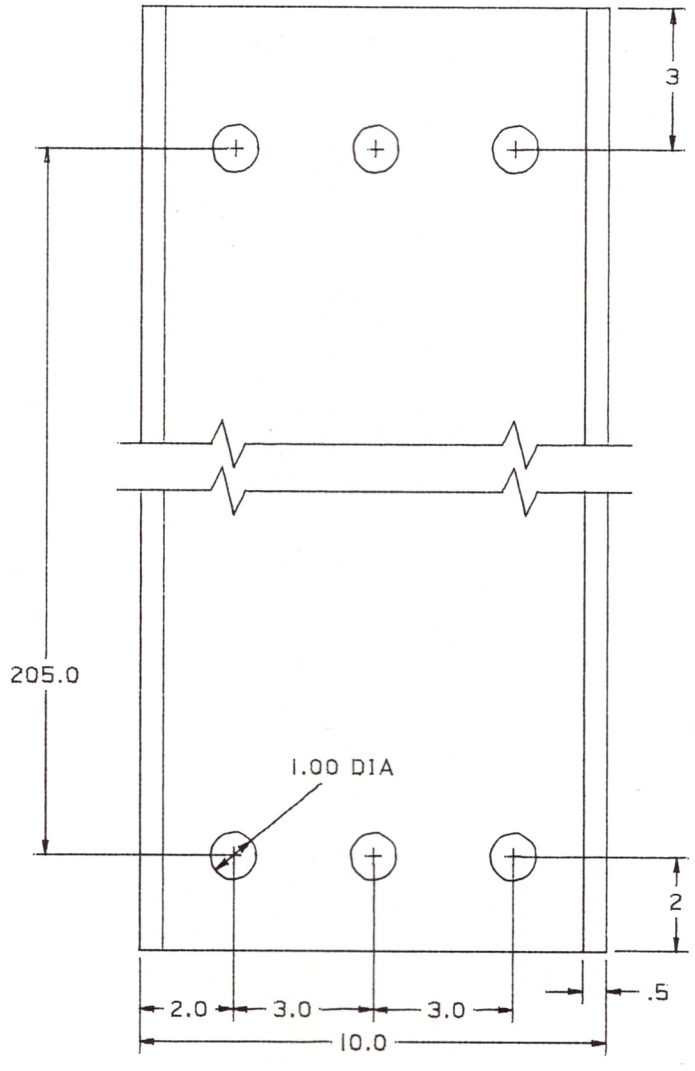
(1) To create large and small scale details on the graphics display screen using the # PAG DIS, # ZOM WIN, # SAV DIS, and # RST DIS commands.
(2) To use the END and ORG masks for entity location.
(3) To insert text on a drawing with the following format.

EXERCISE 4
01-26-83
JANE DOE
JOHN DEERE

A. Create a 210" long wide-flange column as shown below.

B. Insert 1" diameter holes at both ends of the column as shown.

Note that the drawing is not to scale. Do not dimension at this time.



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COMPUTER-AIDED DRAFTING

Unit 5
Plotting

MATCCAD™

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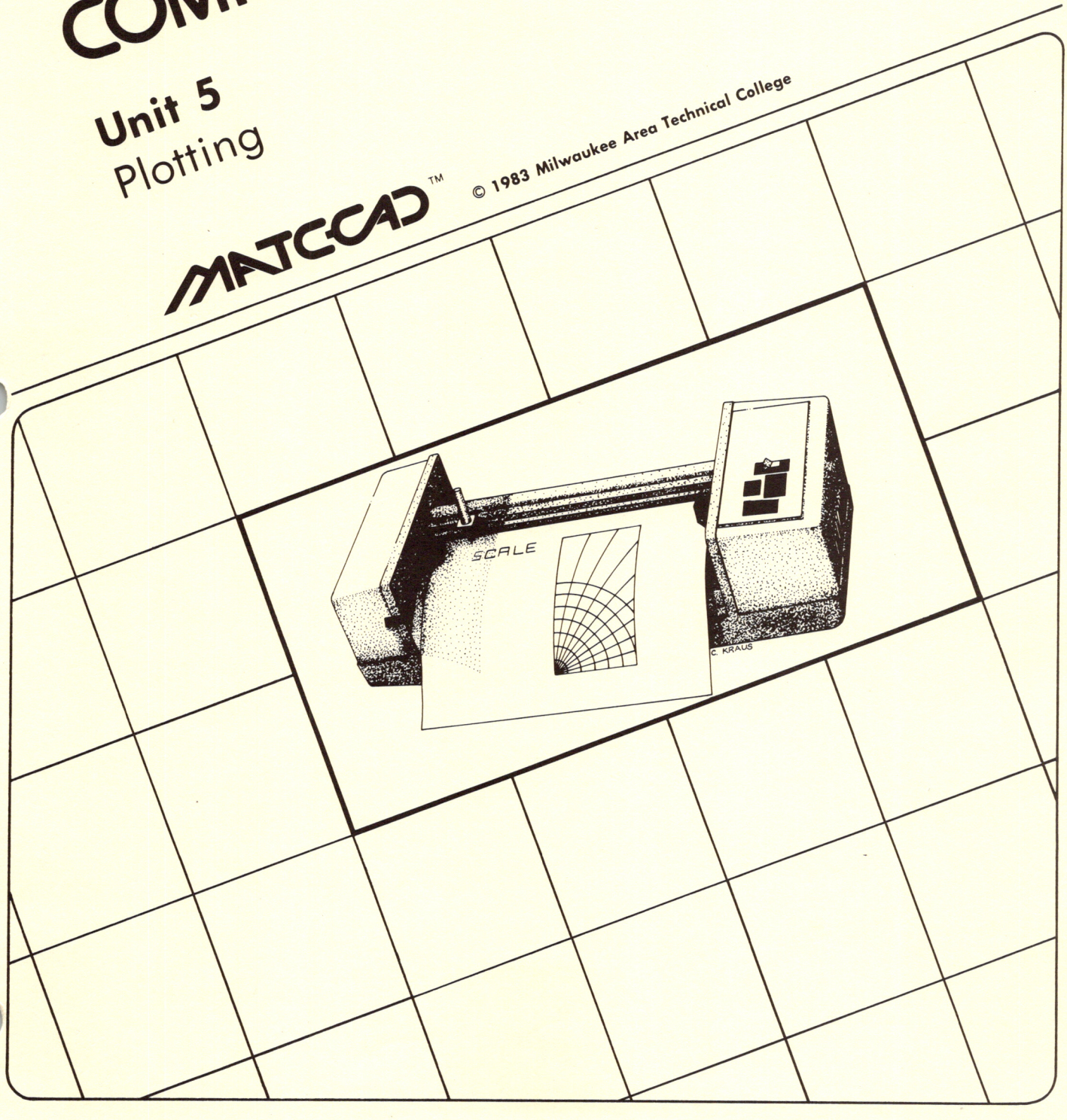

Introduction

One of the final goals of the drafting process is to produce a drawing. This unit will show how to set up and use a pen plotter for producing high quality drawings. This unit also will show how to use a dot matrix printer for quickly obtaining a printed copy of the screen image.

Objectives

At the completion of this unit you will be able to:

1. Use the screen-image scaling and image-shifting commands to place your drawing on the screen at the desired scale and position for subsequent plotting.

2. Set up and use a high-resolution pen plotter for plotting out the drawings you have created.

3. Use a dot matrix printer for printing out a copy of the image on the display screen.

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Unit 5 Reference Section

Part I
The Drawscale Option

Not every part is of a size that lends itself to being plotted out at full scale on a sheet of paper. Whenever buildings, maps, or large assemblies are plotted, they must be drawn at a reduced scale. Likewise, very tiny parts need to be drawn at an increased scale so that the dimensional information can be represented clearly.

A problem arises when very large or small parts are drawn. As the drawing is reduced or enlarged to fill the plotting area, the size of text and dimensioning also shrinks or expands at the same rate, making them too small to see or so large as to obliterate the drawing.

The solution to this problem is for the system to have the ability to scale up or scale down the size of the dimensioning and text so that if a part is plotted at a reduced or enlarged scale, the dimensions and text will plot out at an appropriate size.

On the MATC CAD system, dimensioning and text can be automatically scaled up or down with the drawscale command:

SEL PLOT DSCLn)

The value of "n" gives the scaling factor that is used to enlarge or reduce the displayed size (and also the plotted size) of the text and dimensioning.

As an example, the user has a part with an overall length of around 80 inches and plans to plot the part on A size paper (8 1/2 x 11). The command # SEL PLOT DSCL10) will produce text and dimensioning that are 10 times larger than standard. When the part is then shrunk down to 1/10th of the original size to fit on the paper, the text and dimensions will also shrink down and be plotted out at an appropriate size.

On the MATC CAD system the drawscale can be changed at any time and the text and dimensioning will be rescaled with the next RPNT command. If the plotted text and dimensions are not the desired size, the drawscale can be changed and the drawing plotted again.

The user should note that the drawscale in effect will alter the size of any inserted text. For example, if the drawscale is currently 4 and text is inserted at a specified height of .25 inches, the displayed text will appear four times as large or the same height as a one inch long vertical line on the screen.

On the Computervision system the use of drawscale is different, with the drawscale option following the part name (example: WED.EX8/DSCL.1). The value for the drawscale is also different. # SEL PLOT DSCL10 on the MATC CAD system would be entered as /DSCL.1 on the Computervision system.

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Plotting with MATC CAD

There is a wide selection of plotters available for the graphics market, offering a variety of sizes, costs, speed, accuracy, pen selection, and other factors. Although the MATC CAD system has the potential to work with almost any plotter, the software will be supporting a few plotters that give the best cost performance.

The software that does the plotting for MATC CAD is on a separate disk. To plot your drawing, you will need first to file your drawing on your part disk, then insert the plotting disk into Drive 1 with the MATC CAD Disk 2 in Drive 2. Turn the power on and call up the part to be plotted.

Part II
Selecting the Plotter

The MATCCAD plotting software is designed to allow the user to preview on the screen how the drawing will be plotted. Since each plotter is different in terms of plotting area and operation, it is necessary to tell the computer which plotter and size combination is to be used so that subsequent scaling and zooming commands will result in the correct image being displayed on the screen prior to plotting.

To select the plotter to be used, enter the command: #SEL PLOT)

The computer will then list on the screen the available plotter and size combinations.

Part III
Scaling and Positioning the Part Image for Plotting

The MATC CAD system has been designed to make plotting as easy as possible for the user. Plotting requires only that the user set the desired scale of the plot, position the image within the plotting area, and then plot the drawing.

A. Setting the Plotting Scale

The user has the option of either plotting his/her drawing to a specified scale or scaling the drawing to fill the plotting area. If the user does not want to plot the part to scale, then the command # ZOM ALL will fill the screen and the plotting paper from edge to edge. If a blank border around the part is desired, then # ZOM ALL P10 or # ZOM ALL P20 would be used.

To plot a part to scale, the command #ZOM SCLn is entered where "n" is a screen scaling ratio indicating how large a line will appear in the plotted drawing relative to its actual length in the data base.

This Feature is Present on the Computervision CADDS3® System but has not yet been implemented on MATC CAD.

ZOOM ALL SCL1) Will produce a full-scale plot.

ZOOM ALL SCL2) Will produce a plot twice the actual size.

ZOOM ALL SCL.5) Will produce a plot half the actual size.

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If the drawscale option # SEL PLOT DSCLn is in effect, then the user should choose a plotting scale approximately equal to the inverse of the drawscale. For example, if a user has a part with an overall length of about 160 inches and the final plot is to be on A size paper (8 1/2 x 11) then the user should have used a drawscale value of 20 which will produce text and dimensions 20 times the normal size. When it comes time to plot, the command # ZOOM ALL SCL.05 will shrink the drawing down to 1/20th of the original size, giving an 8-inch-long plotted part and shrinking the text and dimensions down to normal plotted size.

B. Positioning the Image for Plotting

After a # ZOM All SCLn command is entered, the image may not be in the desired position for plotting. If so, the # PAG DIS command should then be used to shift the image on the screen to a different position. Remember that what you see on the graphics screen will be what is plotted out.

C. Plotting the Drawing

Each plotter supported by MATC CAD requires a slightly different procedure at this point and will be explained in the plotter sections that follow.

Note: If you type in #PLOT) without first selecting which plotter you are using with the #SEL PLOT command, the computer will display the NO PLOTTER SELECTED error message.

1. Insert PLOTTER DISK in Drive #1, and MATC CAD DISK 2 in Drive #2

2. Turn power on and procede through the normal LOG-IN and calling up your part to be plotted.

3. The image on the screen can now be positioned to the scale and position desired with a combination of #ZOOM and #PAG DIS commands.

4. The #SEL PLOT DSCLn) command can be used if desired to automatically scale dimensions and text to a more appropriate scale. See page 5-3 in student manual.

5. The plotter to be used and paper size must be selected next. When the command #SEL PLOT) is entered, a list of currently supported plotters and paper sizes will list out on the screen. Select the appropriate # (no RETURN) and you are ready to plot.

6. Insert Paper and pen into you plotter, and when ready enter the command #PLOT).

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Part IV
Hewlett Packard Plotters

HP 7470A A-Size
HP 7475A A/B-Size

The listed specifications for the HP 7470A are:

Size - 10.3 x 7.50 active plotting area
Speed - Up to 15 inches/second
Resolution - 0.001 inches

The listed specifications for the HP 7475A are:

Size - 10.375 x 7.50 or 16.64 x 10.365 active plotting area
Speed - Up to 15 inches/second
Resolution - 0.001 inches

The HP 7470A and HP 7475A can use both felt tip and technical drawing pens, and will produce a very high quality plot.

The procedure for using the HP 7470A or HP 7475A plotter is to:

1. Insert the MATC CAD Plotting Disk into Drive 1 and the MATC CAD Disk 2 into Drive 2.

2. Log-in and load the part you want to plot in the normal manner.

3. Turn the plotter on with the rocker switch on the back.
   The plotter is turned on when the top half of the switch is pushed in.

4. Insert the pen you will be using into the pen holder on the left side.

5. The paper is loaded by first pushing the PAPER LOAD/HOLD lever away from you. Then insert the paper so it is tight against the left rail and rear stop, and then pull the PAPER lever toward you to hold down the paper.

NOTE: Use only the paper supplied to you by your teacher. The plotting paper used with the HP 7470A has a polished surface. If you use standard typing paper the felt-tip plotting pen will wear out very rapidly.

6. Scale and position your drawing on the screen using the ZOOM and PAG DIS commands.

7. Enter the #SEL PLOT) command and select the number corresponding to the 7470A A-size, 7475A A-size, or 7475A B-size.

   There is a speed modifier, # SEL PLOT SPDn), that can be used to set the plotting speed "n" from 1 to 10 (.5 to 15 inches/second).

8. The drawing is then plotted by entering # PLOT)

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Part V
Houston Instruments Plotters

HI DMP-40 A/B-Size
HI DMP-41 C/D-Size

The listed specifications for the HI DMP-40 are:

Size - 9.25 x 7.0 or 14.0 x 9.25 active plotting area
Speed - Up to 4 inches/second
Resolution - 0.005 inches

The listed specifications for the HI DMP-41 are:

This Feature is Present on the Computervision CADDS3® System but has not yet been implemented on MATC CAD.

Size - 20.0 x 15.0 or 32.0 x 20.0 active plotting area
Speed - Up to 3 inches/second
Resolution - 0.005 inches

Both the HI DMP-40 and HI DMP-41 use a felt tip pen, and will produce a high quality plot.

The procedure for using the HI DMP-40 or HI DMP-41 plotter is to:

1. Insert the MATC CAD Plotting Disk into Drive 1 and the MATC CAD Disk 2 into Drive 2.

2. Log-in and load the part you want to plot in the normal manner.

3. Turn the plotter on with the small chrome plated toggle switch located on the back of the plotter. The plotter is turned on when you can see one or more of the red lights glowing on the front touch panel.

4. Insert the pen you will be using into the pen holder.

5. The paper is loaded by first lifting both paper hold down levers so that they point upwards. Next insert your paper so that the left edge of the paper is alligned with the white paper quide line and the bottom edge of the paper is alligned with the front metal edge of the plotter. At this time push both paper hold levers down to lock the paper in position.

NOTE: Use only the paper supplied to you by your teacher. The plotting paper used with the HI DMP-40 has a polished surface. If you use standard typing paper the felt-tip plotting pen will wear out very rapidly.

6. Scale and position your drawing on the screen using the ZOOM and PAG DIS commands.

7. Enter the #SEL PLOT) command and select the number corresponding to the HI DMP-40 A-size, HI DMP-40 B-size, HI DMP-41 C-size, or HI DMP-41 D-size plotter.

   There is a speed modifier, # SEL PLOT SPDn), that can be used to set the plotting speed "n" from 1 to 4.

8. The drawing is then plotted by entering # PLOT)

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Part VI
Screen Image Printing - Epson Dot Matrix Printer

A number of dot matrix printers have the ability to print out a graphic image. The image produced on paper is an exact copy of the image that appears on the screen dot-for-dot. Altough the resolution of the printed image is not that high, it does provide a quick hard copy of your work. Currently the Epson MX 100, MX 80 with Graftrax, RX 80, FX 80, and FX 100 printers are supported.

The procedure for screen image printing consists of:

1. MATC CAD DISK 1 and DISK 2 should be in Drives 1 and 2.

2. Scale and position your part to the desired size and location with the ZOOM and PAGE DIS commands.

3. Enter in the command # PLOT)

The image printed out will be approximately 3x5 inches and compressed slightly vertically due to the unequal vertical and horizontal dot spacing on the Epson printer.

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Unit 5 Assignment Section

Part I
Plotting Exercises

1. If you have a part that is about 30 inches long and want it to fit on 8 1/2 x 11 paper, what would be an appropriate drawscale? _____

2. If you have a part about 1/2" across and want it to fill 1/2 of a plotted page, what would be an appropriate drawscale?

   _____

3. What command will set the scale to plot a drawing at 1/4th of life size?

   _____

4. What command will fill the paper with the part except for a 10% border around the edges? _____

5. What command will print out the screen image of your drawing on an Epson dot matrix printer? _____

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Unit 5 Lab Exercise Section

Lab Exercise 5

Call up the parts _____ and _____ that you have previously created and plot them out using the pen plotter and the dot matrix screen image printer.

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COMPUTER-AIDED DRAFTING

Unit 6
Using Grids
Cartesian and Polar Coordinate Review

MATCCAD™

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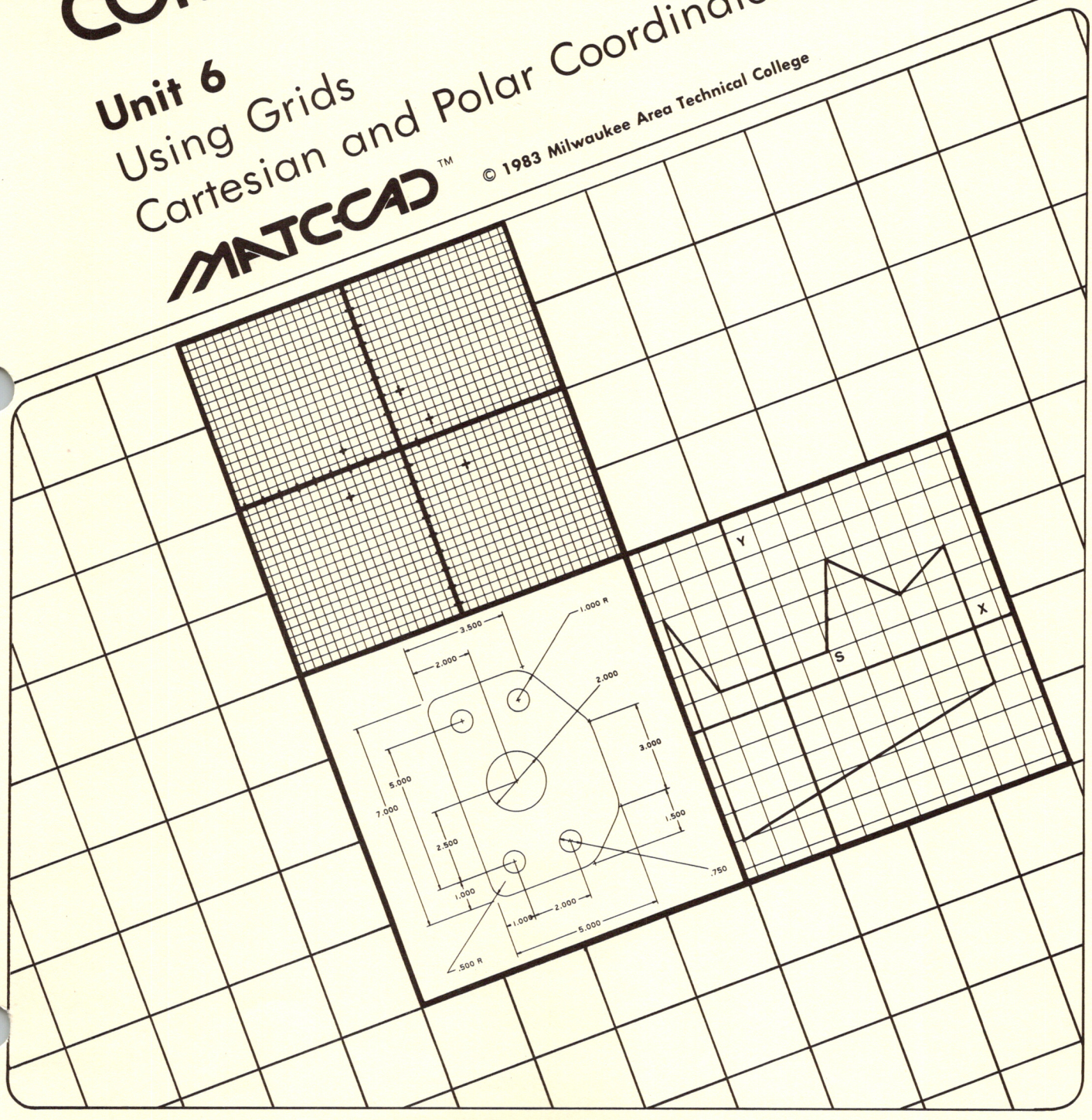

Introduction

The grid feature in a computer graphics system can be used to quickly enter entities in much the same way that graph paper can be used to quickly sketch out the basic outline of a drawing. Grids find use in applications where the designed part is created in a structured spaced manner such as in printed circuit board design. This unit will also review the use of Rectangular (Cartesian) and Polar Coordinates to specify the exact and relative locations for inserting entities. You will use this information in constructing two-dimensional drawings on the computer graphics system.

Objectives

At the completion of this unit you will be able:

1. To use the grid commands as an aid in constructing drawings on the graphics system.

2. When given various existing entities, to specify the command language and coordinate values (cartesian and polar) needed for the construction of these entities.

3. When given the commands and coordinates for the construction of various entities, to draw entities as they should appear on the graphic display.

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UNIT 6 REFERENCE SECTION

Part I
Coordinate Specification

There are two types of coordinate systems that are used on a typical computer graphics system, cartesian coordinates and polar coordinates. The explicit location of entities (such as lines, points, and circles) can be specified using either coordinate system in either of two ways:

1. Absolute Coordinates - With absolute coordinates, data locations are defined in relation to their position from the origin (X0Y0).

2. Incremental Coordinates - Using incremental coordinates, data locations are defined in relation to the position of the last entered location.

A. Cartesian (Rectangular) Coordinates

In the cartesian coordinate system the position of a point, the end of a line, etc. is located or specified by a pair of X Y values (coordinates). These two values describe a given location in terms of its horizontal and vertical distance from some fixed point of reference called the "origin." The coordinates of the origin itself are X=0 and Y=0.

A drawing of a typical XY coordinate system is shown below. Note that the X value will give the horizontal (left-right) value of the position, while the Y value gives the vertical (up-down) value of the position.

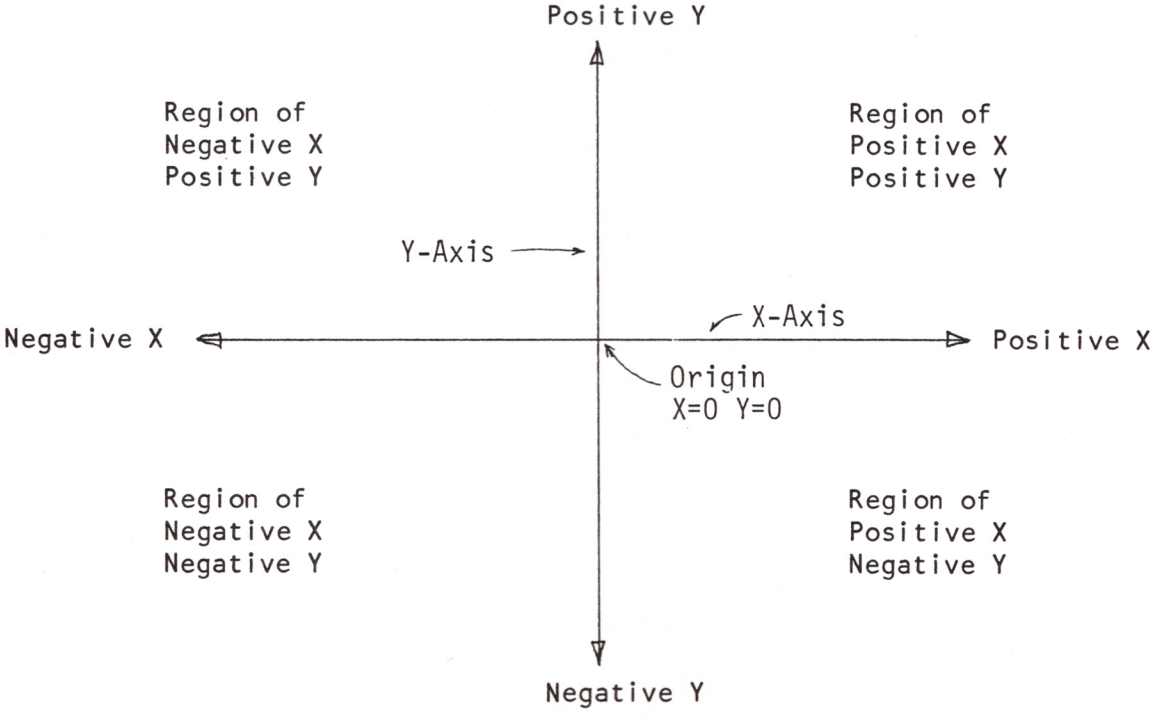

Figure 1 - Cartesian coordinate system

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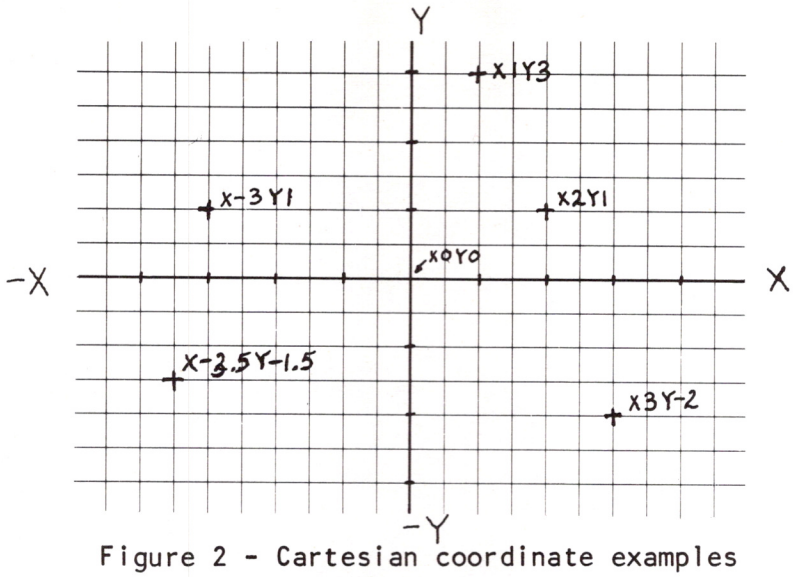

Figure 2 - Cartesian coordinate examples

On the MATC CAD graphics system the origin is initially positioned at the center of the screen. A point at the position X=3, Y=5.2 (3 inches to the right and 5.2 inches above the origin) would be specified as X3Y5.2. A point at X=0, Y=-3.6 would be specified as X0Y-3.6.

If you want to specify a location in terms of its position in relation to an existing location, this would be done with incremental coordinates. For example, if you just finished specifying a point at X5Y10, then the location at X6Y12 could be specified by the incremental coordinates IX1 IY2. (That is, you are increasing the current X position by 1 and the current Y position by 2).

Absolute Cartesian Coordinates

X and Y are used as the position specifiers for rectangular absolute coordinates.

X = The distance along the horizontal axis away from the origin.
Y = The distance along the vertical axis away from the origin.

Incremental Cartesian Coordinates

IX and IY are used to specify the rectangular incremental coordinates.

IX = The incremental horizontal distance from the previous position.
IY = The incremental vertical distance from the previous position.

B. Polar Coordinates

In the polar coordinate system a position is determined by the direct distance of the position from the origin in combination with the angle of the position from the origin measured in a counterclockwise direction from the horizontal. Negative angles would be clockwise from the horizontal. The following figure shows a typical polar coordinate system with some polar coordinate locations shown.

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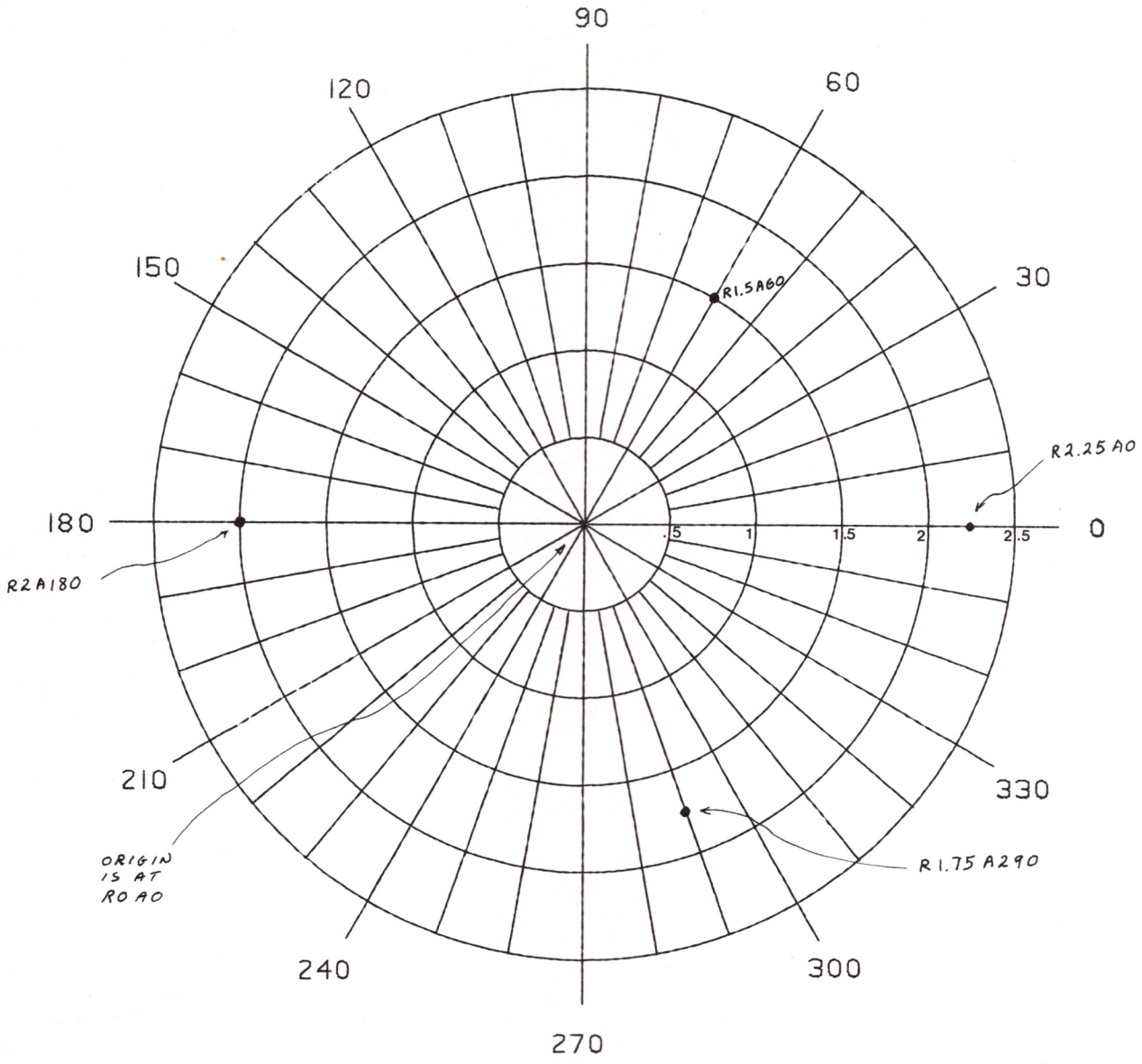

Figure 3 - Polar Coordinate System With Examples

Absolute Polar Coordinates

The position specifiers for absolute polar coordinates are R for the distance from the origin and A for the angle from the horizontal.

A = Angle and R = Radius

Incremental Polar Coordinates

The position specifiers for incremental polar coordinates are IR for the distance from the last position specified and IA for the angle of the new position based on the current direction of the previous line in the same command.

IA = Incremental Angle IR = Incremental Radius

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B. Explicit Input

1. Basic Command

Explicit coordinates are specified following the colon in the Data Input part of the command. Each explicit coordinate defining a position is separated by a comma.

Data input section

INS LIN : DIG X0Y0, X3Y4)

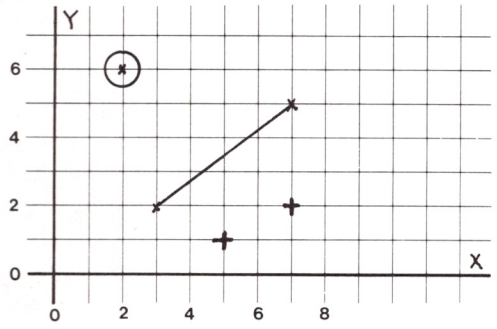
2. Explicit coordinate input can be used to specify the ends of a line or the centers of circles and the location of points.

INS LIN : DIG X3Y2, X7Y5)

Beginning of line.
End of line.

INS CIR R.5 : DIG X2Y6)

INS PNT : DIG X5Y1, X7Y2)

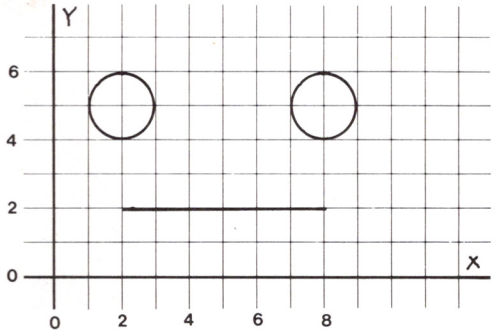


3. An X or Y coordinate value that does not change from the previously specified value in the command may be left out and assumed to have the same value as before.

INS LIN : DIG X2Y2, X8)

(The Y value of 2 did not change so it can be left out)

INS CIR R1 : DIG X2Y5, X8)



Note that the sequence of entering X or Y (or A or R) makes no difference. X2Y3 is equivalent to Y3X2. R2A30 is equivalent to A30R2.

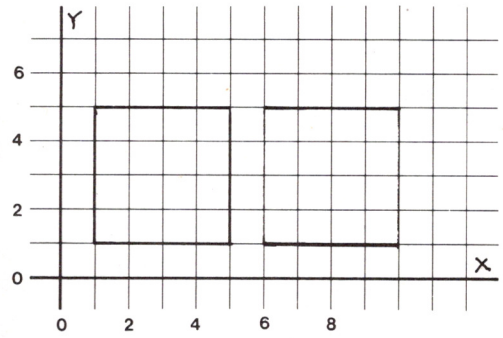
4. When inserting lines, a comma signals the completion of the explicit coordinate specification for a position and a semi-colon causes a reset for a new line.

INS LIN : DIG X1Y1, X5, Y5, X1, Y1)

all commas

INS LIN : DIG X6Y1, X10; DIG X10Y5, X6)

(the semi-colon starts a new line)



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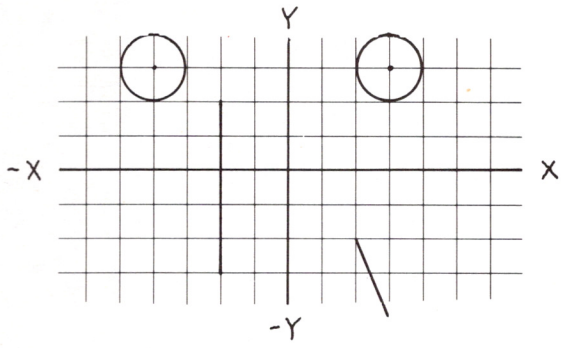
5. The value for a coordinate can be either positive or negative with negative numbers indicated by a minus sign.

INS LIN : DIG X-2Y2, Y-3)

INS LIN : DIG X2Y-2, X3Y-4)

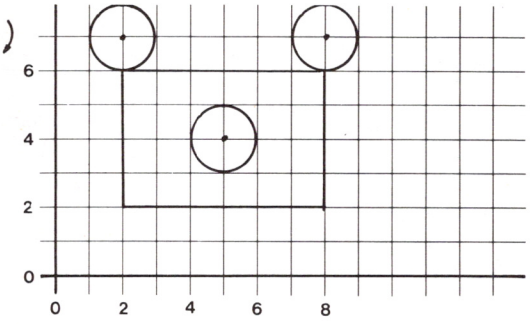
INS CIR R1 : DIG X3Y3,X -4)



6. Incremental coordinates are specified in relation to the last position entered and can be intermixed with absolute coordinates.

INS LIN : DIG X2Y2, IX6, IY4, IX-6, IY-4)

INS CIR R1 : DIG X2Y8, IX6, X5Y4)

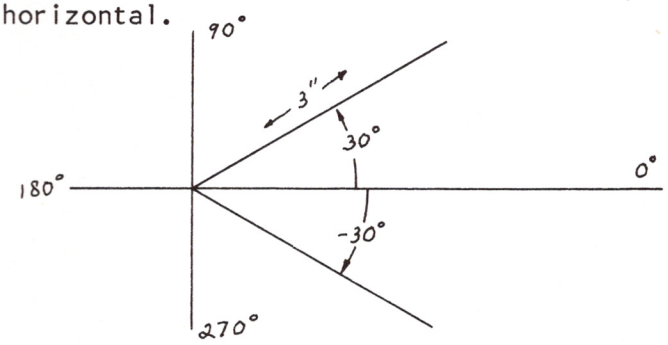


7. The same rules apply for specifying location of entities using polar coordinates as for rectangular coordinates. The horizontal axis pointing to the right is 0 degrees and all absolute angles are determined by counterclockwise rotation from the horizontal.

INS LIN : DIG X0Y0, R3 A30)

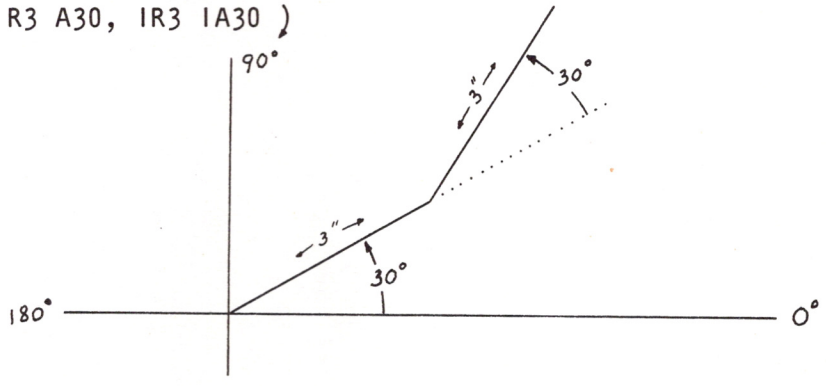
INS LIN : DIG X0Y0, R3 A-30)

With lines, R indicates the length of the line.



8. An incremental angle is always based on the angle of the last line entered in the same command. Incremental angles will not work if there was not a prior line specified in the command.

INS LIN : DIG X0Y0, R3 A30, IR3 IA30)



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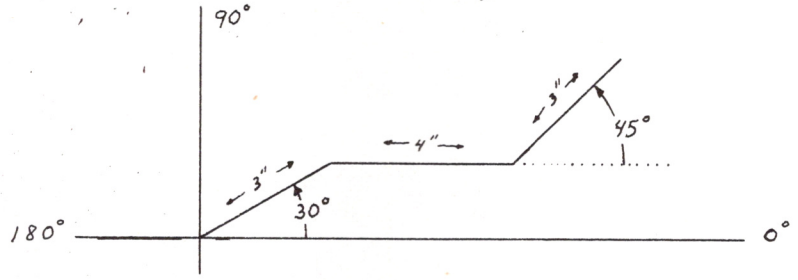
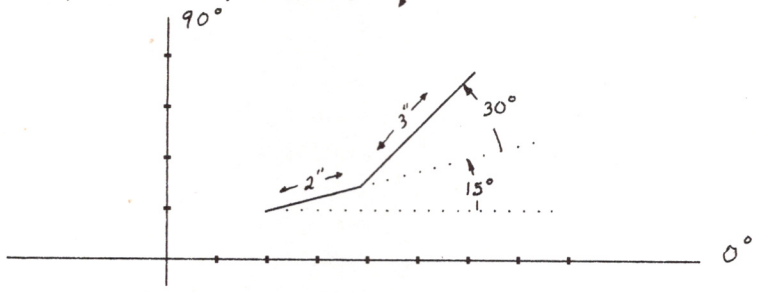
9. Polar, rectangular, absolute, and incremental coordinates may be intermixed.

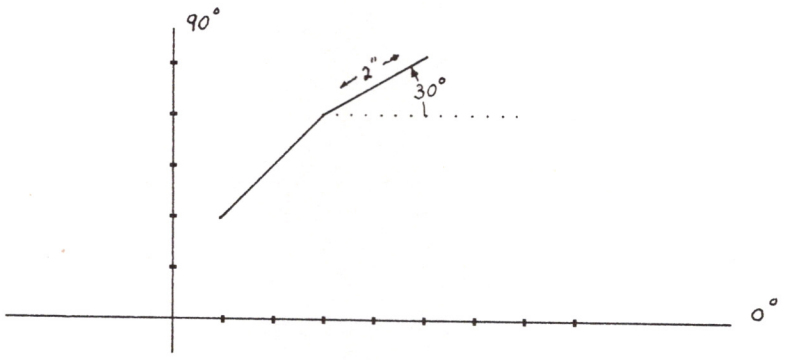
INS LIN : DIG X0Y0, A30 R3, IX4, IR3 A45)



INS LIN : DIG X2Y1, IR2 A15, IR3 IA30)



INS LIN : DIG X1Y2, X3Y4, IR2 A30)



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Part II
Using Grids

Grids are used as a guide for the non-critical location of data positions. They are also convenient for locating text and dimensions.

A. Basic Grid Command

SEL GRD (modifiers))

Modifiers

Gn Sets the horizontal and vertical spacing. The default spacing is .5.
ON Turns grid on (activates) *
OFF Turns grid off (deactivates) *

* When the grid is selected ON, any digitizing done with the digitizing pen will snap to the nearest grid point.

To specify the origin of the grid use the ORG modifier followed by a colon. The origin of a grid is typically set to the END or ORG of an entity, or explicit XY coordinates can be used for the origin location.

This Feature is Present on the Computervision CADDS3® System but has not yet been implemented on MATC CAD.

SEL GRD ORG : DIG X7Y9)
RPNT)

B. To Make Grid Visible (and Invisible)

ECH GRD ON) Note: A grid does not have to be visible to be active.
RPNT)

ECHO GRD OFF) To make the grid invisible (invisible is the default)
RPNT)

C. Typical Grid Command Sequence

The typical sequence of commands needed to set up a grid are:

1) # SEL GRD G.5) Specify the grid size (.5" size for this example)

2) # SEL GRD ON) Turn grid on. (Activate) Any digitizing will snap to the nearest grid point.

3) # ECH GRD ON) To make the grid visible.

4) # RPNT) A repaint will then display the visible grid.

D. List Grid Command

The command LIST GRID) is used to find out the size and active status of the currently defined grid.

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UNIT 6 EXERCISE SECTION

Part I
Grid Exercises

Directions: Write the language as it would appear on the command display to do the following exercises.

Example	Command Language
Make a previously specified and selected grid visible on the graphics display.	# ECH GRD ON) # RPNT

1. Specify a grid with an X distance of .5 and Y of .5 visible on the screen, but be able to input data by coordinates rather than by snapping to grid points.

   1. _____

2. Specify a grid with an X and Y distance of 2.0 and be able to use the grid to insert entities. Don't have the grid visible.

   2. _____

3. Specify a deactivated grid with an X and Y distance of .25. The grid originates at X0Y0 and is visible on the graphics display.

   3. _____

4. Specify a grid with an X and Y distance of .75. The grid originates at X2Y2.

   4. _____

5. Find out whether a grid is on or off. (Activated or deactivated)

   5. _____

6. Find out the current grid spacing.

   6. _____

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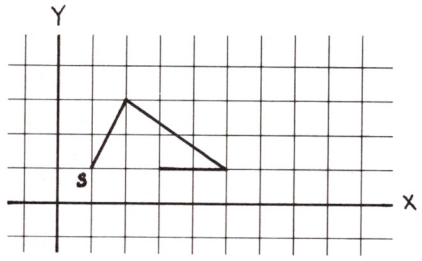
Part II
Cartesian Coordinates

Directions: Using cartesian coordinates, give the command that would be needed to contruct the following figures on a graphics design station. NOTE: On any of the exercises below, "s" will indicate a starting point for the command.

Example



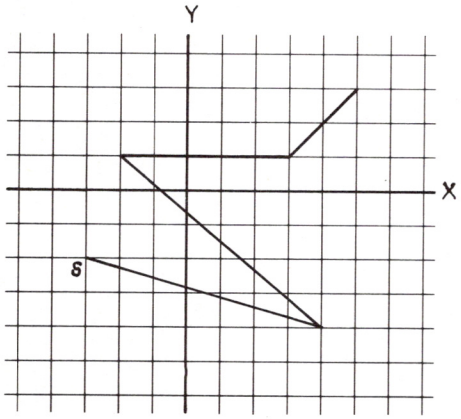
Command Language

INS LIN : DIG X1Y1, X2Y3, X5Y1, X3)

or

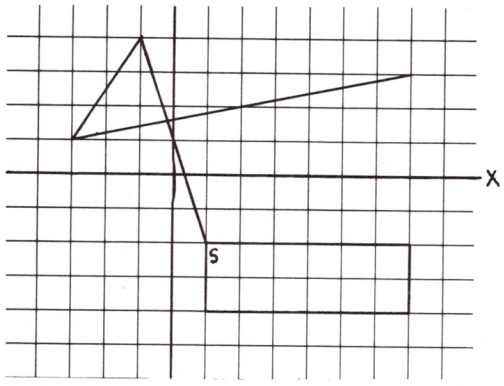
INS LIN : DIG X1Y1, IX1IY2, IX3IY-2, IX-2)

1.



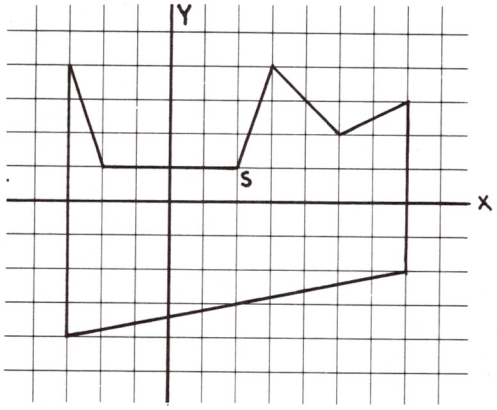
1.

2.



2.

3.



3.

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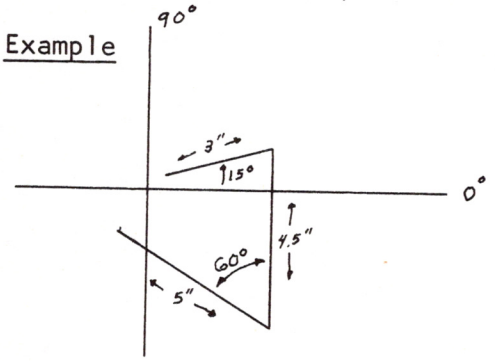
Part III
Polar Coordinates

Directions: Write the command language as it would appear on the command display that would be needed to construct the following examples. Use Polar coordinates wherever possible.

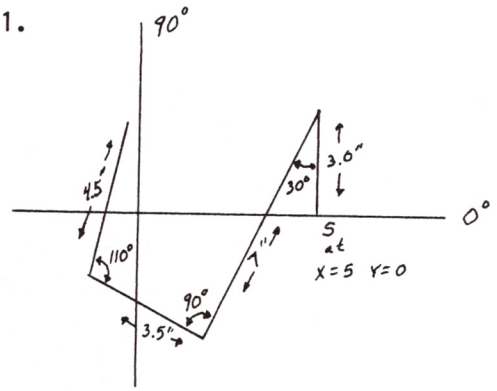
Example



Command Language

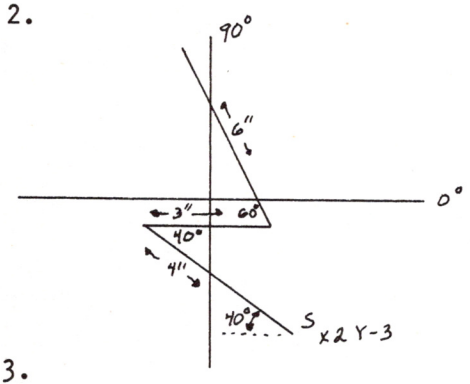
INS LIN : DIG X.5Y.5, IR3A15,
IR4.5A270, IR5IA240)

1.



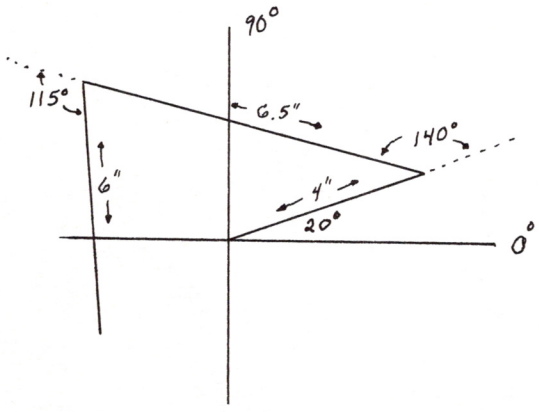
1.

2.



2.

3.



3.

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Directions: Sketch the entities generated as a result of the following commands:

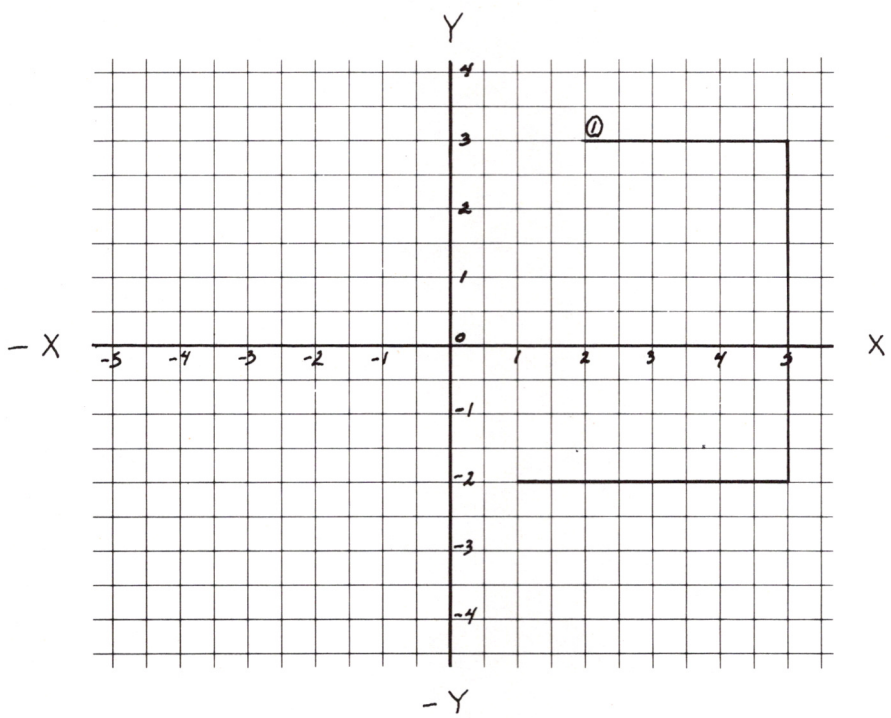
Example # INS LIN : DIG X2Y3, X5, Y-2, X-4)

A. # INS LIN : DIG X-5Y1, IX1Y3; X-3Y3, IX2, IX-2IY-2)

B. # INS LIN : DIG X-2Y-2, IX1IY1, Y-3, IR3A180)

C. # INS LIN : DIG X1Y1, IX2, IY-1, X4Y2)

D. # INS LIN : DIG X0Y0, R2A-45, X1Y-3, IR2A0, IR1A-90, R4A-80)



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UNIT 6 LAB EXERCISES

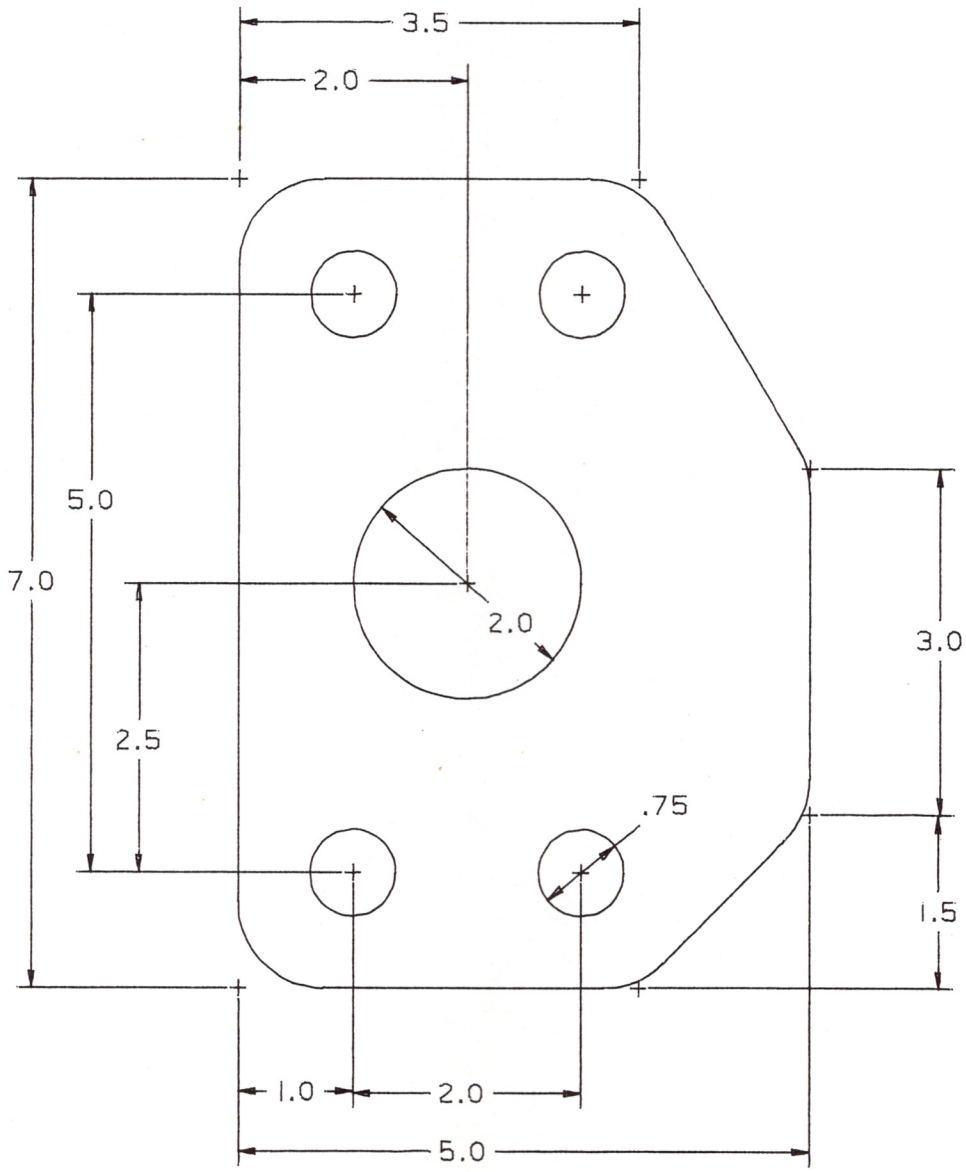
Directions - Construct the following drawings. Do not dimension at this time.

Use the part name _____ for Lab Exercise 6A.

Use the part name _____ for Lab Exercise 6B.

Upon completion have your instructor check them.

LAB EXERCISE 6A



ALL CORNERS .75 RADIUS

USE .5 GRID FOR INITIAL WORK

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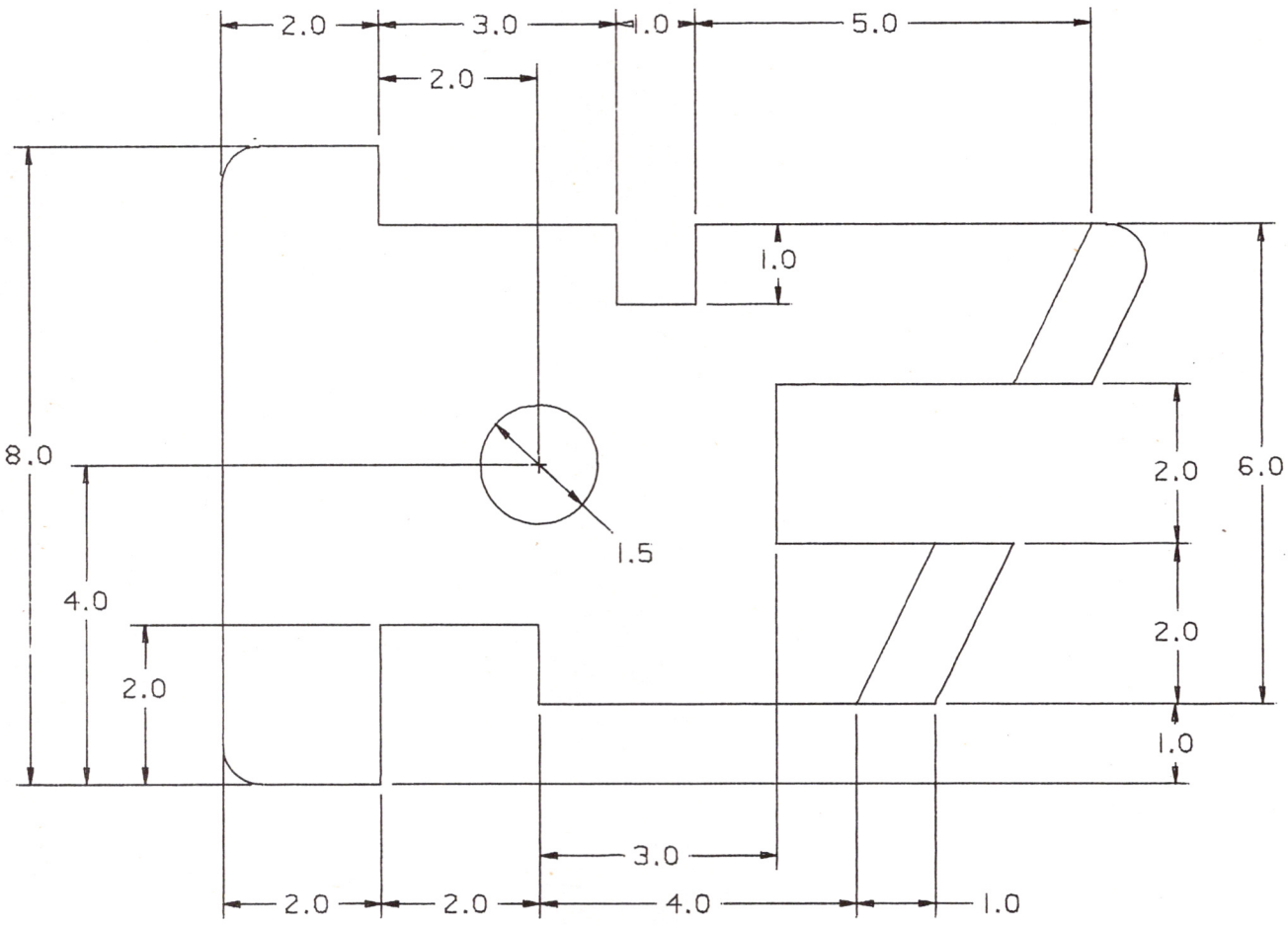
LAB EXERCISE 6B

Do not dimension at this time.



CORNERS 0.5 RADIUS

USE 1.0 GRID FOR INITIAL WORK

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COMPUTER-AIDED DRAFTING

Unit 7
Lines

MATCCAD™

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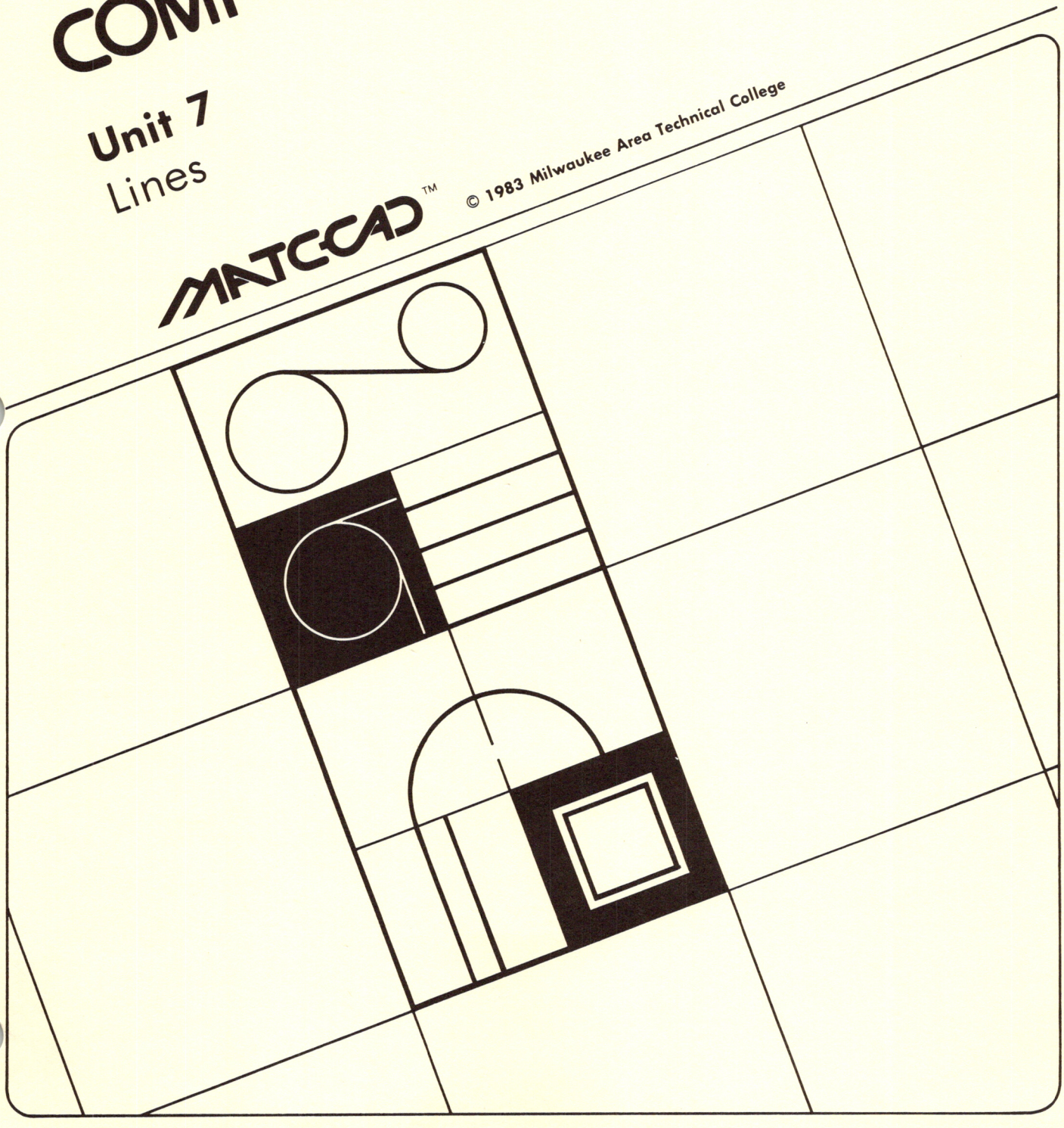

Introduction

There are typically many ways in which entities such as lines, circles, and arcs can be created. This unit will explain the advanced methods used in constructing lines. This information will be used as a aid in the construction of two-dimensional drawings on the graphics design system. Various types of line styles can be used on a computer graphics system. This unit will also give instruction and practice in the commands to select a particular line style (SEL LFNT) and change a line to another style (CHG LFNT).

Objectives

At the completion of this unit you will be able:

1. To give the command language needed to construct horizontal, vertical, parallel, and perpendicular lines, lines tangent to circles, and lines of a specified length and/or angle.

2. When given various lines, to write out the commands (as they would appear on the display) and show the digitizes that would be needed to create these various entities.

3. Given the commands that would create various lines, to draw these entities as they would appear on the graphic display screen of a graphics design station.

4. Specify different line fonts using the SEL LFNT command.

5. Write the commands that would be used to change the line fonts of existing entities using the CHG LFNT command.

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Unit 7 REFERENCE SECTION
Lines

In addition to inserting lines between two locations specified by digitizing or explicit coordinates, there are a number of modifiers that can be used with the Insert Line command to specify exactly how you want the line to be created in terms of its length, angle, tangency, or parallel distance from an existing line.

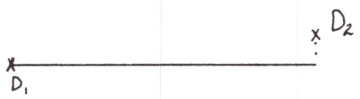
These modifiers are:

VER	Forces the line to be inserted vertically.
HOR	Forces the line to be inserted horizontally.
ANG n	To draw a line at an angle "n" to an existing line.
LNG n	To draw a line of a specified length "n".
PRP	To draw a line perpendicular to a selected existing line.
PRL n	To draw a line parallel to a selected line. The parallel distance can be specified with "n".
TANTO	To draw a line tangent to an existing circle.

Note: When using the modifiers horizontal (HOR), vertical (VER), perpendicular (PRP), and parallel (PRL), the length of the line you are creating is determined by dropping a perpendicular from the location of the last digitize in the command to the new line specifying where the new line will end.

Part I
Horizontal and Vertical Lines

INS LIN HOR : DIG D_1 D_2)



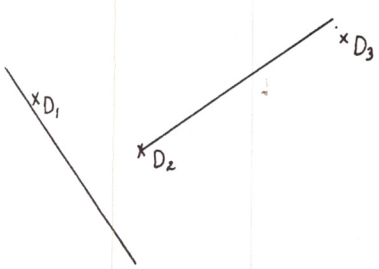
D_1 Specifies the start of the line.
D_2 Gives the horizontal extent of the line.

INS LIN VER : DIG D_1 D_2)

D_1 Specifies the start of the line.
D_2 Gives the vertical extent of the line.

Part II
Perpendicular Lines

INS LIN PRP : ENT D_1 DIG D_2 D_3)



D_1 Selects the entity to be perpendicular to.
D_2 Specifies the beginning of the line.
D_3 Will specify the "end" of the created line.

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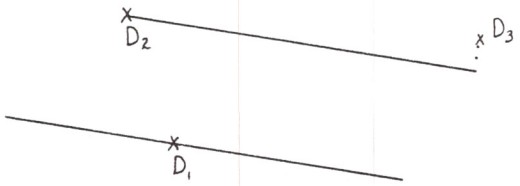
Part III
Parallel Lines

There are two methods of inserting parallel lines. With the first method (using three digitizes), the second digitize specifies the starting position and therefore the parallel distance. In the second method (using four digitizes), the parallel distance is specified by a number "n" following the modifier PRL.

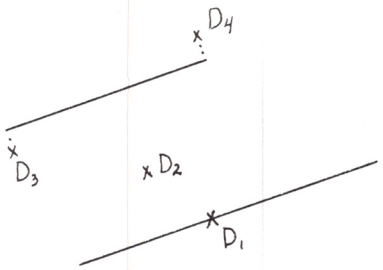
INS LIN PRL : ENT D_1 DIG D_2 D_3)

D_1 Selects the entity to be parallel to.
D_2 Indicates starting point, the parallel distance, and which side the created line is on.
D_3 Indicates the "end" of the new line.



INS LIN PRL 3 : ENT D_1 DIG D_2 D_3 D_4)

D_1 Selects the entity to be parallel to.
D_2 Indicates which side the new line will be on.
D_3 Indicates the "starting point" of the new line using a perpendicular dropped to the parallel distance of 3 inches.
D_4 Indicates the "end" of the new line.



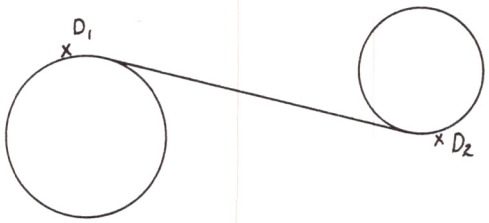
Part IV
Tangent Lines

There are three ways to insert a line that is tangent to an existing circle. In each case the circles should be selected at a location close to the expected point of tangency for the line to be created.

1. A line tangent to two circles.

   # INS LIN TANTO : ENT D_1 D_2)

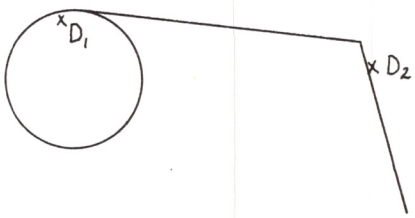
   Digitize the sides of the circles that the line is to be tangent to.



2. A line tangent to a circle and touching the end of a line or a point.

   # INS LIN TANTO : ENT D_1 END D_2)

   Used when you want a line tangent to a circle but through an existing point or the end of a line.



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3. A line tangent to a circle and either parallel or perpendicular to another existing line.

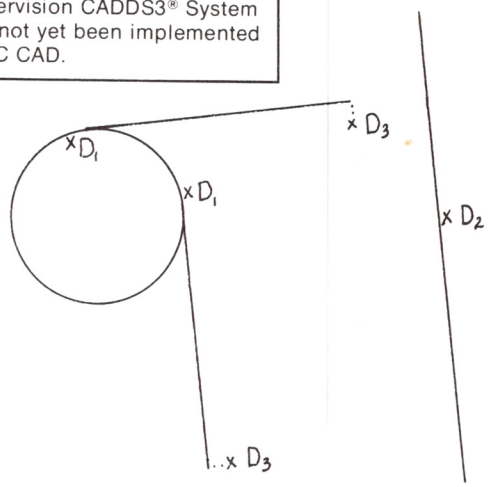
This Feature is Present on the Computervision CADDS3® System but has not yet been implemented on MATC CAD.

INS LIN TANTO : ENT D_1 D_2 DIG D_3 ↵

D_1 Selects the circle to be tangent to and approximate point of tangency.

D_2 Selects an existing line to be parallel or perpendicular to.

D_3 Indicates the extent or end of the created line. The created line will be either parallel or perpendicular to the line selected with D_1 depending on the location where the circle entity was selected.

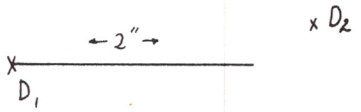


Part V
Line of a Given Length

The LENGTH (LNGn) modifier when used with the INS LIN command will explicitly specify the length of a line constructed. LNG must be used with one of the modifers HOR, VER, ANG, PRP, or PRL.

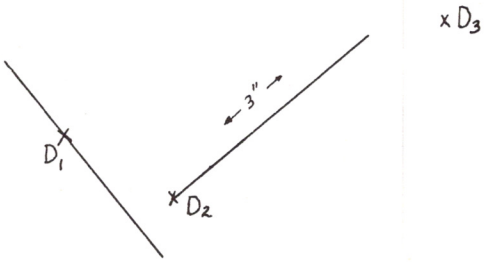
INS LIN HOR LNG 2 : DIG D_1 D_2 ↵

D_1 Gives the beginning location of the line.
D_2 Gives direction of construction (to the left or right for the horizontal line in this example).



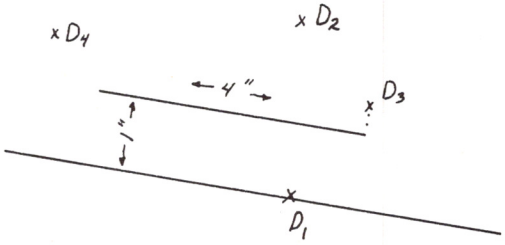
INS LIN PRP LNG 3 : ENT D_1 DIG D_2 D_3 ↵

D_1 Identifies the entity that the new line is to be perpendicular to.
D_2 Gives the beginning of the new line.
D_3 Gives direction of construction of the new line.



INS LIN PRL 1 LNG 4 : ENT D_1 DIG D_2 D_3 D_4 ↵

D_1 Identifies the entity that the new line will be parallel to.
D_2 Picks on which side of the line the parallel line will be created.
D_3 Gives the starting end of the created line.
D_4 Gives the direction of the created line from D_3 .



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Part VI
Line at a Given Angle to an Existing Line

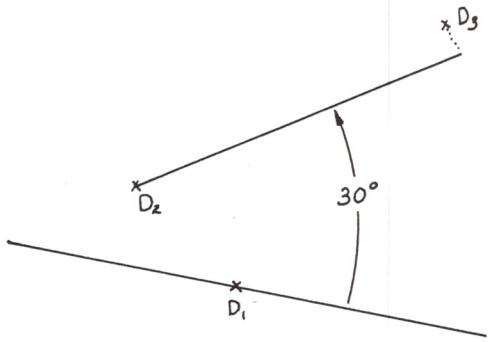
The ANGLE (ANG) modifier when used with the command INSERT LINE inserts a line at a specified angle "n" with respect to an existing selected line.

INS LIN ANG 30 : ENT D_1 DIG D_2 D_3)

D_1 Identifies the existing line to be used as the reference for the new line created.

D_2 Gives the starting location of the created line.

D_3 Gives the direction and the "end" of the newly created line.



Part VII
Line Fonts

Selecting a Line Font Type

The default type for a line font is solid. Other fonts are available for selection. The following fonts affect lines, circles, and arcs.

To change the line font parameter:

SEL SOLID) all lines created from this point will be solid

SEL DASH) all lines created from this point will be dashed

For ease in recognizing dashed lines on the display screen, the MATC CAD software is designed to display the dash length at a constant screen size regardless of the screen scale of the drawing. When dashed line is plotted however, it will vary in length with the scale of the drawing.

Once a line font is selected, all lines will be of that type until a different line font is selected.

Example:

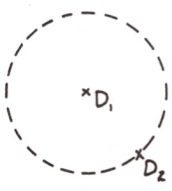
SEL DASH)

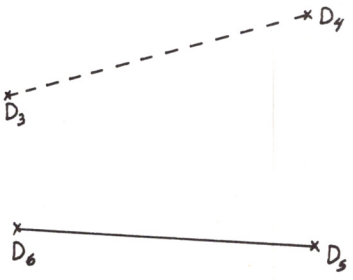
INS CIR R : DIG D D)

INS LIN : DIG D D)

SEL SOLID)

INS LIN : DIG D D)





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Changing the Font of a Line

The commands to change the font of specific lines to a different font are:

CHG LFNT SOLID : ENT D - D) the selected entities are changed to solid

CHG LFNT DASH : ENT D - D) the selected entities are changed to dash

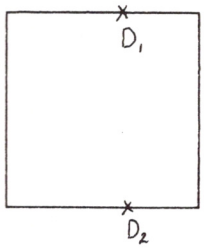
Data input masks such as WIN, LIN, ARC, etc., may be used as an aid in the selection process.

Example:

CHG LFNT DASH : ENT D_1 D_2)

RPNT)

Before



After

Because it takes longer to generate and display dashed lines on the screen, it is recommended that you insert all lines and circles as solid, and then change to dash when your drawing is completed.

Part VIII
Centerlines

This Feature is Present on the Computervision CADDS3® System but has not been implemented on MATC CAD.

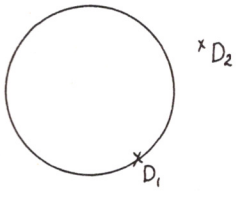
INS CLIN : DIG D D)

INS CLIN CIR : ENT D DIG D)

example:

INS CLIN CIR : ENT D DIG D)

Before



After

An alternate method for generating centerlines which produces excellent results is to set up a small-sized grid centered on the object you want to centerline. Then just create your own centerlines by going grid point to grid point with lines.

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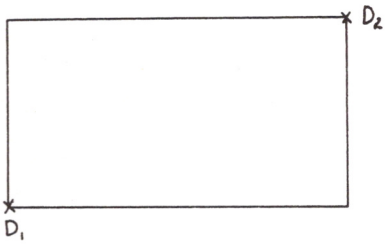
Part IV
Inserting Rectangles

The rectangle insertion command (#INS REC) allows the user to create a rectangle by indicating the coordinates of two opposing corners of the rectangular figure. The first corner should be entered as an absolute XY coordinate value. The coordinate of the opposing corner can be specified by entering either :

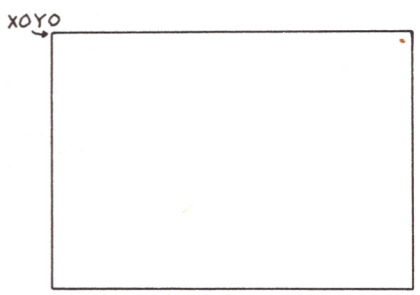
1. The absolute XY coordinate value
2. The incremental XY coordinate relative to the first corner position
3. Using an incremental polar coordinate relative to the first position

INS REC : DIG D_1 D_2)

D_1 Location of one corner
D_2 Location of corner opposite D_1



INS REC : DIG XOYO, IX3IY-2)



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UNIT 7 EXERCISE SECTION

Part I
Line Exercises

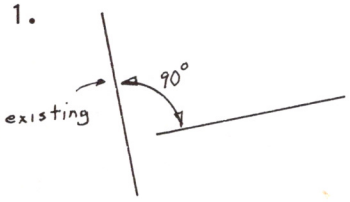
Directions: Write the language as it would appear on the command display and indicate the digitizes that would be needed to construct the following:

example:

Command Langauge

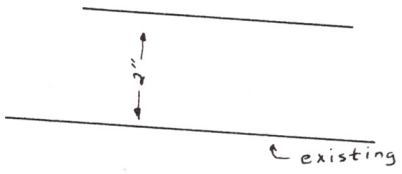
INS LIN HOR : DIG D_1 D_2)

1.



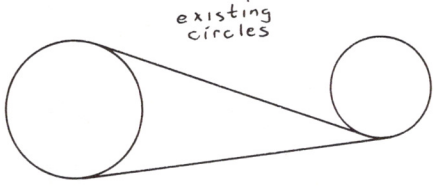
1.

2.



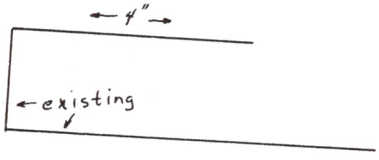
2.

3.



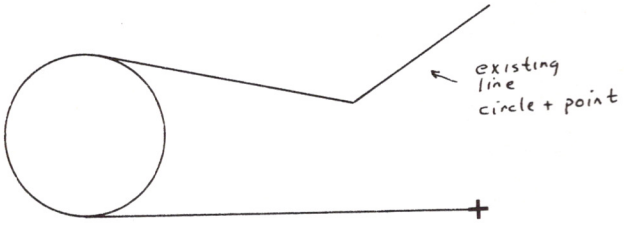
3.

4.



4.

5.



5.

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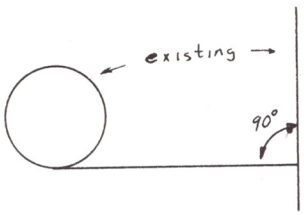
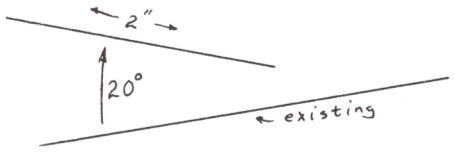
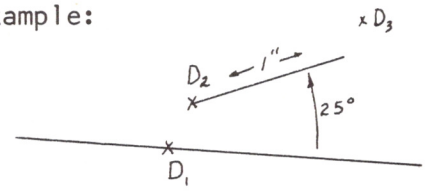
6. 

6.

7. 

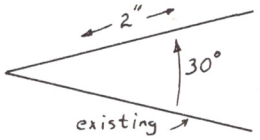
7.

Directions: Give the commands and digitizing locations necessary to construct the following exercises.

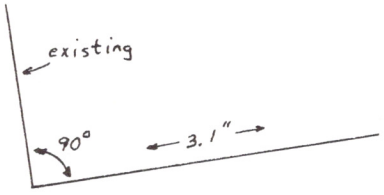
example: 

Command Language

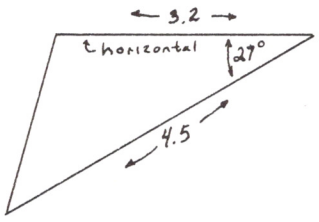
INS LIN ANG 25 LNG 1 : ENT D_1 DIG D_2 D_3)

1. 

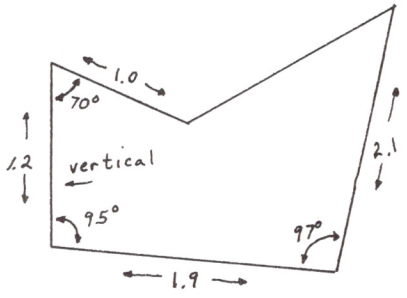
1.

2. 

2.

3. 

3.

4. 

4.

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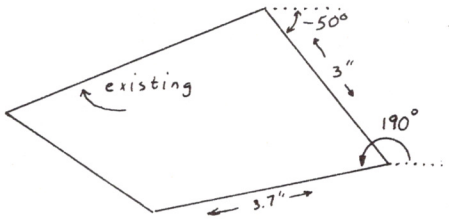
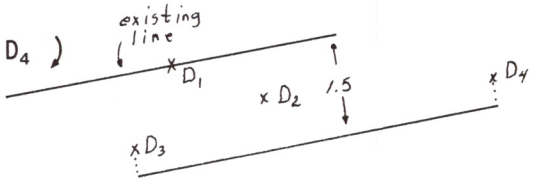
5.



5.

Directions: Construct the lines called for by the following language. Insert the digitizes and dimensions as needed.

example: # INS LIN PRL1.5 : ENT D_1 DIG D_2 D_3 D_4)

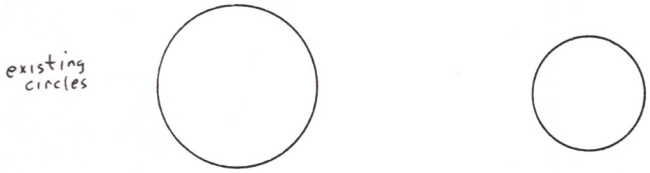


1. # INS LIN PRL2 : ENT D_1 DIG D_2 D_3 D_4)

2. # INS LIN PRP : ENT D_1 DIG D_2 D_3)

3. # INS LIN PRL : ENT D_1 DIG D_2 D_3)

4. # INS LIN TANTO : ENT D_1 D_2)



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Part II
Line Font Exercises

Directions: Answer the following questions:

1. If I wanted all entities to be drawn as dash lines on the graphics display, what command would be entered?

   _____

2. If I wanted to change some entities to dashed lines on the graphics display, what command would be entered?

   _____

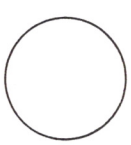
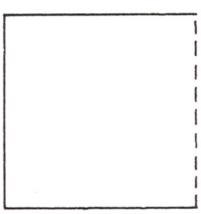
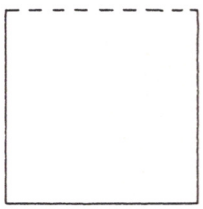
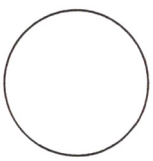
3. What is the default line font style?

   _____

4. If all entities are being drawn on the data display as dashed lines, what command would be entered on the command display to return to a solid line font?

   _____

Directions: Write the command language as it would appear on the graphics display and show the digitizes that would be used to achieve the following modifications.

Before	After	Command Language
example:		# CHG LFNT DASH : ENT D)
_____	- - - - - - - - -	
1. 		1.
2. 		2.
3. 		3.

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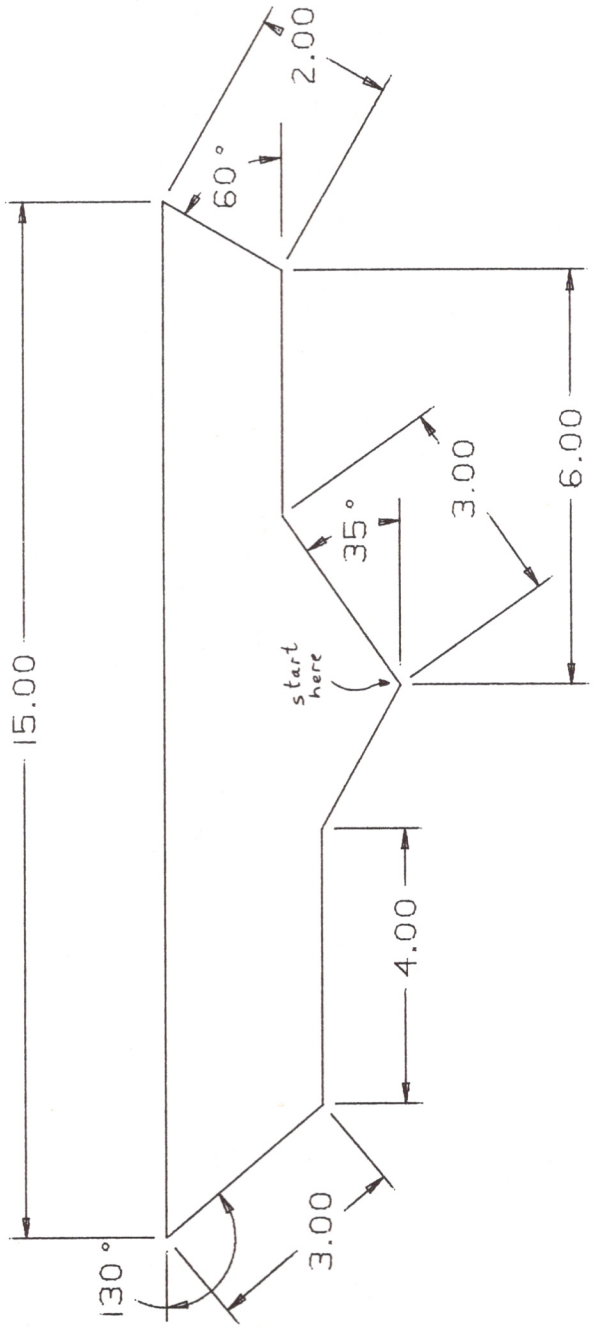
Unit 7 Lab Exercise

Directions: Construct the following drawings on the graphics display. Use the part name _____ for Exercise 7A and the part name _____ for Exercise 7B. Upon completion, have your instructor check it. Do not dimension the part at this time.

Lab Exercise 7A

Do not dimension at this time.



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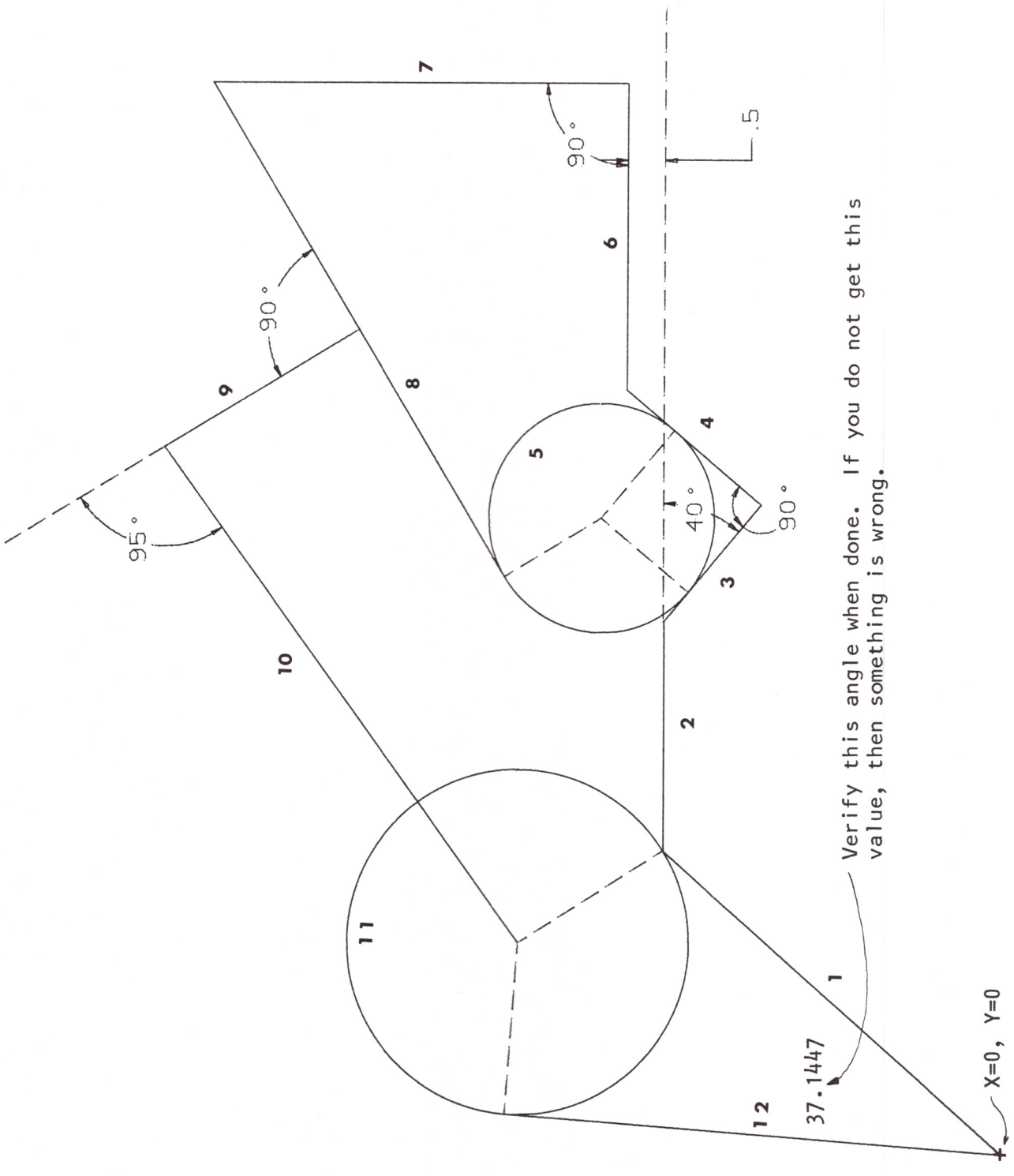
Lab Exercise 7B



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Lab Exercise
Construction Information

Directions: To be done without the use of trigonometry or a calculator.

1. Line starting at (0,0) with length of 6" and the end 4.5" up. (Hint, the line will end at the intersection of a line and a circle.)
2. 3" horizontal line.
3. 2" line at -40 degree angle.
4. Line of unknown length perpendicular to line 3.
5. Circle with 3" diameter that touches lines 3 and 4 . (Hint, locate the center to insert the circle, or use a fillet.)
6. 4" horizontal line parallel to line 2 but .5" higher.
7. 5.5" vertical line.
8. Line tangent to circle 5 and touching the end of line 7.
9. 3" line perpendicular to line 8 from the midpoint origin of line 8.
10. 8" line at a 95 degree angle from the end of line 9.
11. Circle of unknown radius with center at the end of line 10 and circumference touching the intersection of lines 1 and 2 .
12. Line tangent to circle 11 with other end at origin (0,0).

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COMPUTER-AIDED DRAFTING

Unit 8
Inserting Circles, Arcs, Points, and Fillets

MATCCAD™

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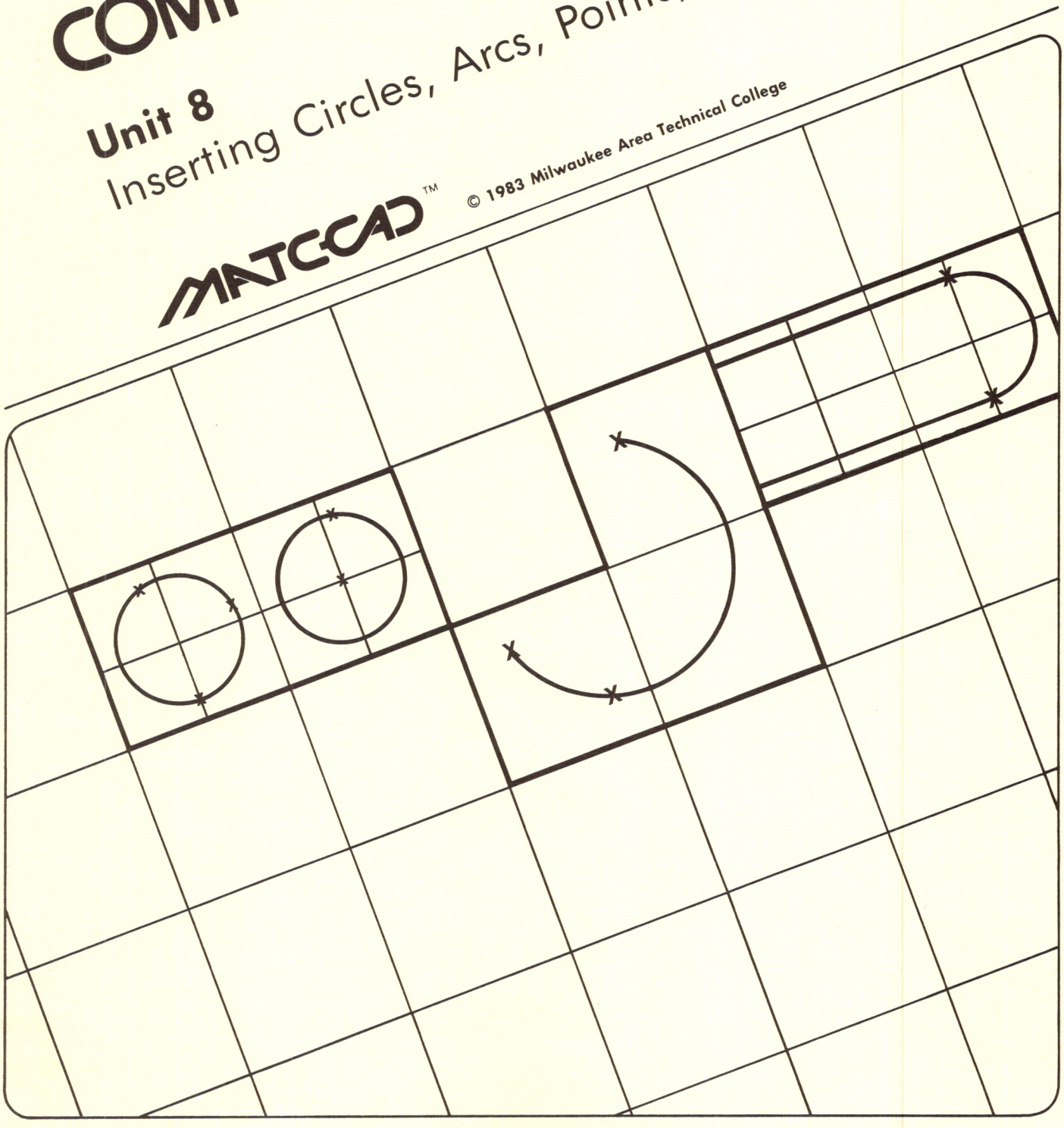

Introduction

There are many ways in which circles, arcs, points, and fillets can be constructed and inserted. This unit will explain advanced methods used in constructing circles, points, fillets, and arcs. This information will be used as an aid in the construction of two-dimensional drawings on the graphics design system.

Objectives

At the completion of this unit you will be able:

1. To give the command language needed to construct circles, arcs, and fillets of a specified or unspecified radius.

2. When given various circles, points, and fillets, to write out the command (as it would appear on the display) and show the digitizes that would be needed to create these various entities.

3. Given the command language that would create various circles, fillets, and points, to draw these entities as they would appear on the graphic display screen of a graphics design station.

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UNIT 8 REFERENCE SECTION

Part I
Inserting Circles

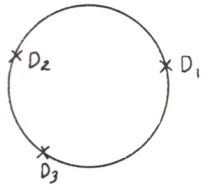
The Insert Circle command can insert circles in six different ways. A circle is created in a counterclockwise direction with its start and end at zero degrees (3 o'clock position). A circle's origin is at its center.

Insert Circle Modifiers	
(none)	The circle is drawn through three specified locations.
R	The radius is specified through digitizing.
Rn	The radius is specified by giving its value following R.
D	The diameter is specified through digitizing.
Dn	The diameter is specified by giving its value following D.

A. Circle Through Three Locations

Digitize three points, which will be on the circumference of the circle.

INS CIR : DIG D_1 D_2 D_3)

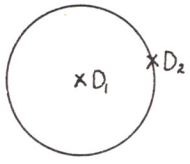


B. Circle of Unspecified Radius

Using the modifier R with no radius given, digitize the center and a point somewhere on the circumference.

INS CIR R : DIG D_1 D_2)

D_1 Center of circle.
D_2 Somewhere on the circumference.

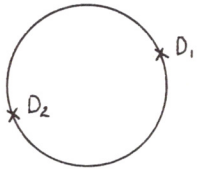


C. Circle of Unspecified Diameter

Using the modifier D, digitize two points which will be the diameter.

INS CIR D : DIG D_1 D_2)

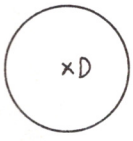
The origin (center) of the resulting circle is located midway between D and D



D. Circle with a Specified Radius

Explicitly specify the radius or diameter and digitize the center of the circle.

INS CIR R2.5 : DIG D)



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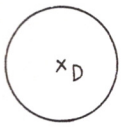
E. Circle with a Specified Diameter

Explicitly specify the diameter and digitize the center point.

INS CIR D4.25 : DIG D)

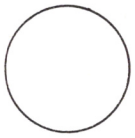


F. Changing an Arc to a Circle

Any arc or fillet can be changed to a full circle with this command.

INS CIR ARC : ENT D)

Before

After 

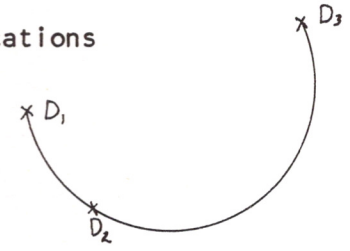
This Feature is Present on the Computervision CADDS3® System but has not yet been implemented on MATC CAD.

Part II
Inserting Arcs

Although there are thirty-one ways in which arcs can be inserted on a Computervision system, most arcs can be generated using only a few methods. Remember that like circles, arcs are always generated in a counterclockwise direction.

A. Inserting an Arc Through Three Locations

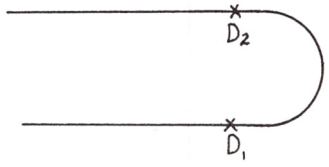
INS ARC : DIG D_1 D_2 D_3)



B. Inserting a Semicircle (180 degree Arc)

This is done using the modifier D without a specified value. In the following example an arc is inserted at the end of two parallel lines to form a rounded slot. The END mask is entered by the user.

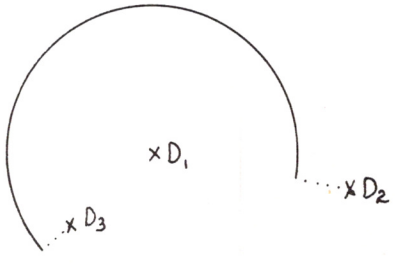
INS ARC D : DIG END D_1 D_2)



The semicircle will go counterclockwise from D_1 to D_2 .

C. Inserting an Arc With a Specified Radius

INS ARC R3.5 : DIG D_1 D_2 D_3)



D_1 Is the arc center.
D_2 Specifies the beginning angle of the arc.
D_3 Specifies the ending angle of the arc.

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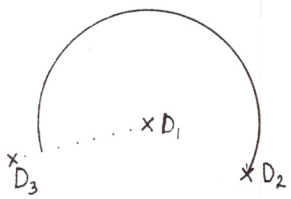
D. Inserting an Arc With an Unspecified Radius R

INS ARC R : DIG D_1 D_2 D_3)

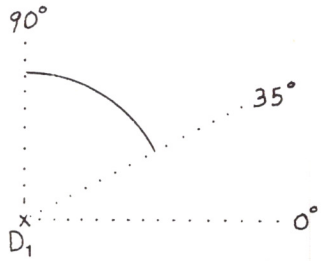


D_1 Is the center of the arc.
D_2 Specifies the radius and the beginning angle of the arc.
D_3 Specifies the ending angle of the arc.

E. Inserting an Arc With the Radius, Beginning and Ending Angles Specified

The modifiers AGOn and AENDn are used to specify the beginning and ending angles.

INS ARC R.75 AG035 AEND90 : DIG D)



D_1 Indicates the center.

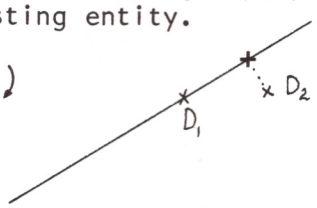
Part IV
Inserting Points

A point is a non-dimensional entity occupying a single X Y coordinate. It may be inserted anywhere in the part and used as a reference or insertion point for other entities. It is indicated on the graphic display as a plus sign (+) of constant screen size regardless of screen scale.

Insert Point Modifiers

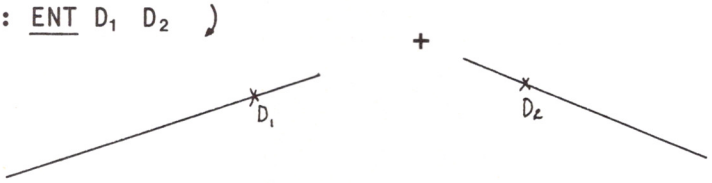
ON - With the modifier ON, a point will be inserted on the selected entity or an extension of the entity by dropping a perpendicular from the digitized location to the selected existing entity.

INS PNT ON : ENT D_1 DIG D_2)



INTOF - With the modifier INTOF, a point will be inserted at the intersection of two lines or curves selected by the user.

INS PNT INTOF : ENT D_1 D_2)



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Part V
Inserting Fillets

A fillet is an arc inserted at the intersection of two lines in the same plane. Fillets are typically used to round off the inside or outside of a corner. If the two lines do not intersect, a fillet will be inserted at their projected intersection.

Important - A fillet, like a circle or arc, is formed in a counterclockwise direction. Therefore, in digitizing entities to form a fillet, the selection **must** be so that the fillet will be formed in a counterclockwise direction. After the fillet has been inserted, the computer will automatically trim the lines to the fillet ends.

Insert Fillet Modifiers

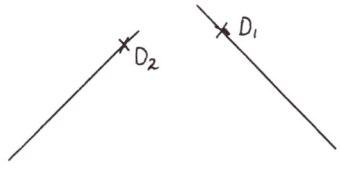
Rn	to specify the radius of the inserted fillet
Dn	to specify the diameter of the inserted fillet
-TRIM	will leave lines unchanged after fillet is inserted

This Feature is Present on the Computervision CADDS3® System but has not yet been implemented on MATC CAD.

If no modifier is used you will get a default radius of .5"

```
# INS FIL : ENT D1 D2 )
# RPNT )
```

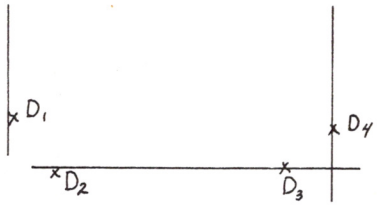
Before



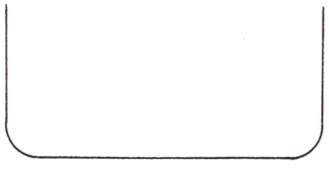
After

```
# INS FIL R1 : ENT D1 D2 ENT D3 D4 )
# RPNT )
```

Before

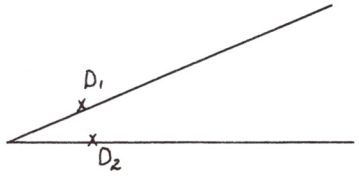


After

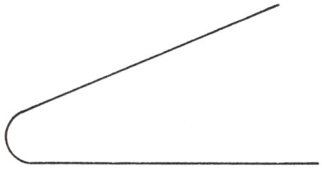


```
# INS FIL R1.5 : ENT D1 D2 )
# RPNT )
```

Before

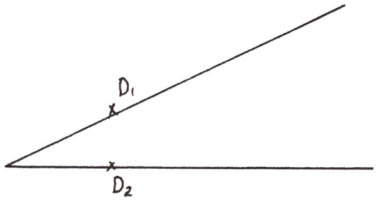


After

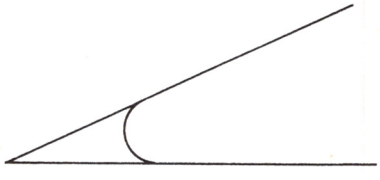


```
# INS FIL R2 -TRIM : ENT D1 D2 )
# RPNT )
```

Before



After



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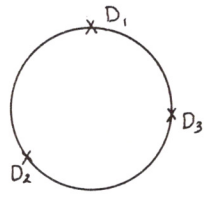
UNIT 8 EXERCISE SECTION

Part I
Circle Exercises

Directions: Give the commands needed to construct the following circles.

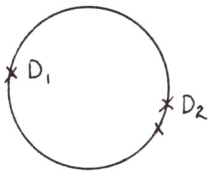
example:



Command Language

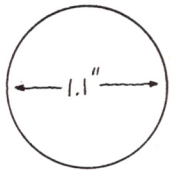
INS CIR : DIG D_1 D_2 D_3)

1.



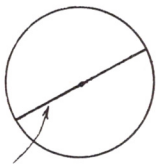
1.

2.



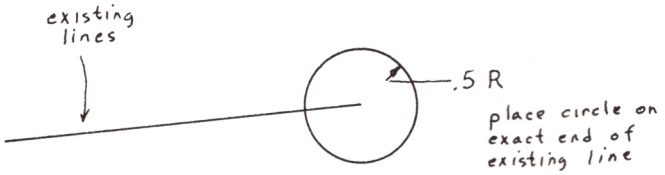
2.

3.



3.

4.

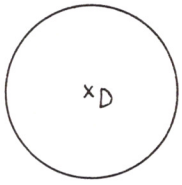


4.

Directions: Construct the circles called for by the following commands. Indicate digitize locations as needed.

example:

INS CIR R1.2 : DIG D)



1. # INS CIR : DIG D_1 D_2 D_3)

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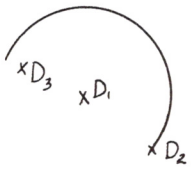
2. # INS CIR R : DIG D_1 D_2)

3. # INS CIR D : DIG D_1 D_2)

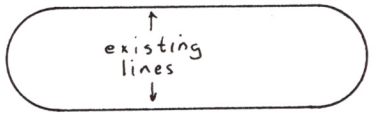
4. # INS CIR D2.5 : DIG D)

Part II
Arc Exercises

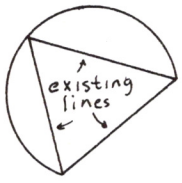
Give the commands needed to construct the following arcs.

1. 

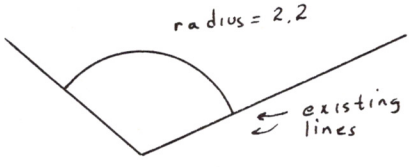
1.

2. 

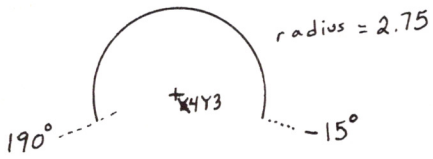
2.

3. 

3.

4. 

4.

5. 

5.

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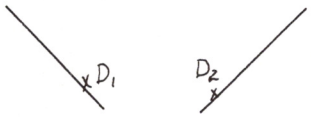
Part III
Fillet Exercises

Directions: Write the command and indicate the digitizing locations that would be needed to construct the following:

Example:

Before

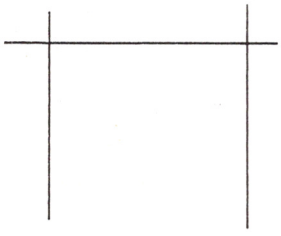


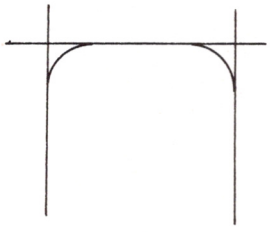
After

Command Language

INS FIL : ENT D_1 D_2)
RPNT)

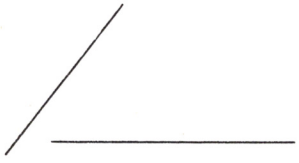
1.

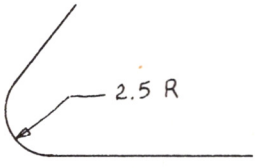




1.

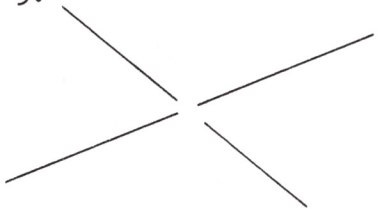
2.

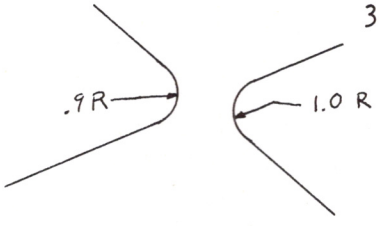




2.

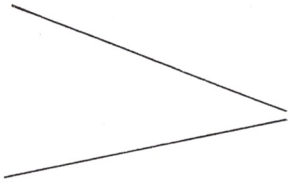
3.

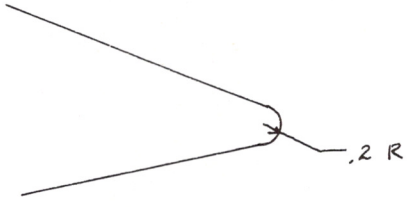




3.

4.





4.

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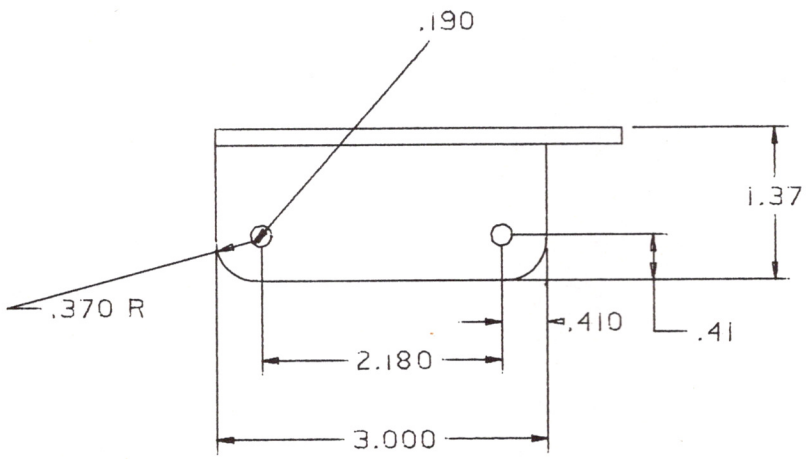
UNIT 8 LAB EXERCISES

Directions: Construct the following drawings on the computer graphics system.

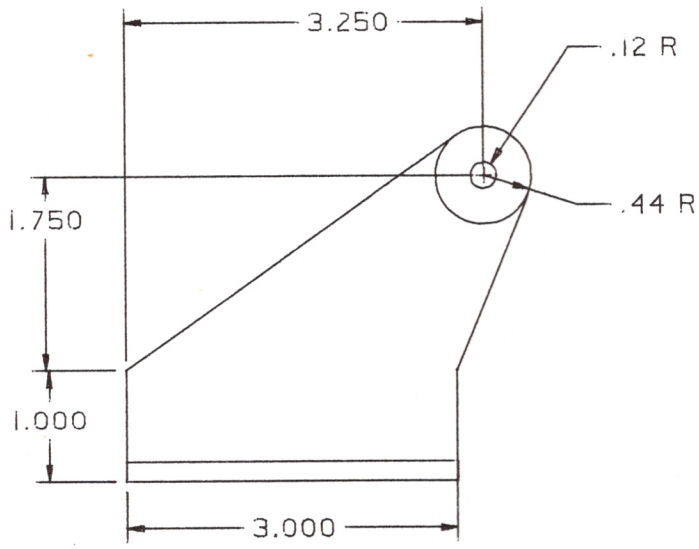
Use the part name _____ for Lab Exercise 8A

Upon completion, have your instructor check them.

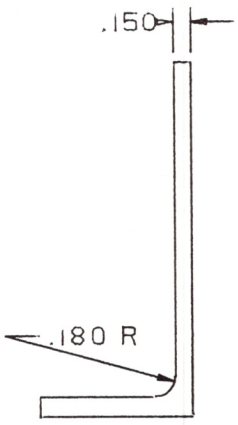
Lab Exercise 8



VIEW 3



VIEW 1



VIEW 2

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COMPUTER-AIDED DRAFTING

Unit 9
Entity Modification

MATCCAD™

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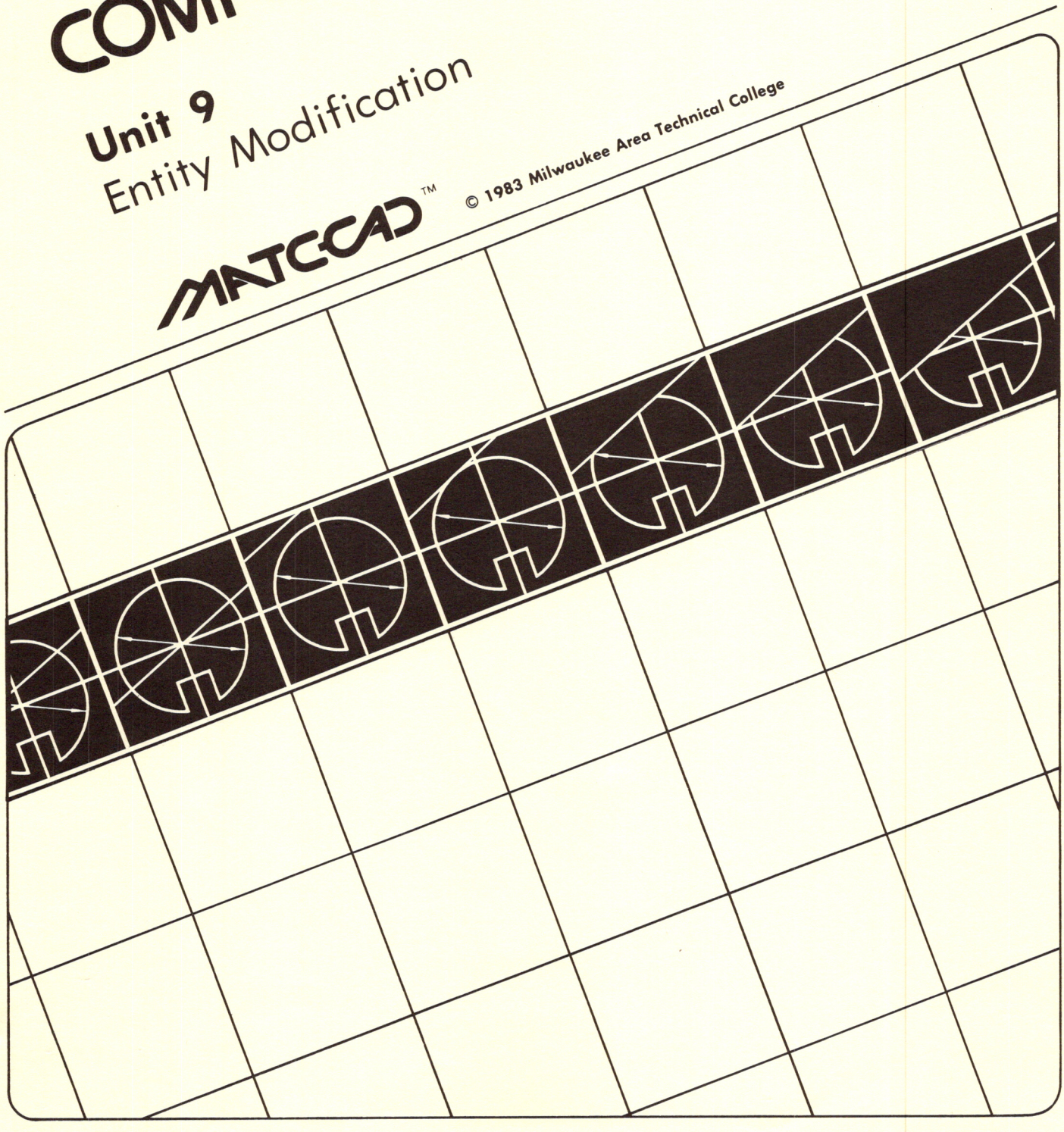

Introduction

This unit will give you practice in modifying existing lines and circles through the use of the entity modification command (TRIM), and give you instruction and practice in breaking up an entity into several smaller segments of the same type with the aid of the entity division command (DIV ENT). You will also practice changing the line font or erasing portions of an entity that has been divided by using different modifiers with this command. This information will be used to construct and dimension drawings using the computer graphics system.

Objectives

At the completion of this unit, you will be able to:

1. Use the TRIM command with the intersection modifier (INTOF) to relocate the length of existing lines and circles.

2. Write the commands needed to divide various entities into specific portions using the DIV ENT command.

3. Write the commands needed to change the line font or erase portions of entities using the DIV ENT command with appropriate modifiers.

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Lab Exercise 9B	

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UNIT 9 REFERENCE SECTION

Part I
Trim Command

The TRIM command allows you to relocate the ends and thereby the length or angles of existing lines, circles, fillets, and arcs. When a line, circle, or arc is to be modified, the user has the option to either trim both ends of the entity to be modified or to just trim one end. If only one end is to be trimmed, the computer will trim the end closest to where the entity was digitized.

A. Basic Command

TRIM (noun) (modifier) : ENT D DIG D D)

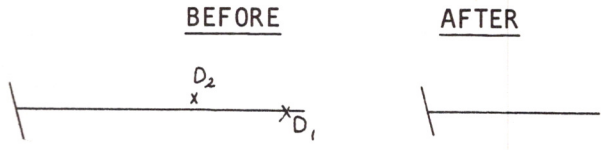
B. Trimming Lines

TRIM LIN : ENT D_1 DIG D_2)

D_1 Identifies the line and which end of the line is to be trimmed.
D_2 Identifies new location of end of line to be trimmed.

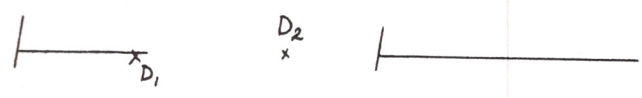
1. Trimming to shorten a line.

TRIM LIN : ENT D_1 DIG D_2)



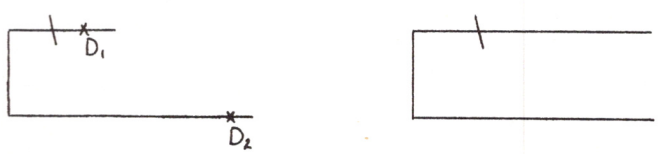
2. Trimming to extend a line.

TRIM LIN : ENT D_1 DIG D_2)



3. Using the end of an exising line or point as a reference for trimming.

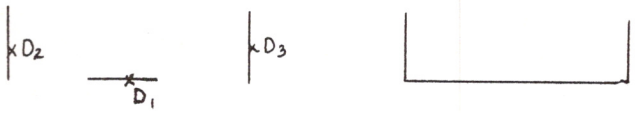
TRIM LIN : ENT D_1 DIG END D_2)



4. Trimming using the modifier INTOF.
Allows trimming of an entity at its intersection with another entity.
When the modifier INTOF is used the prompt ENT will appear instead of DIG.

TRIM LIN INTOF : ENT D_1 ENT D_2)

TRIM LIN INTOF : ENT D_1 ENT D_2 D_3)
(In this example, both ends are trimmed longer).



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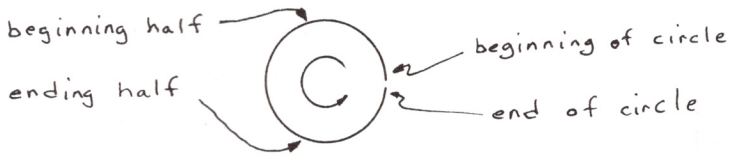
5. Trimming to a specified length.
The modifier LNGn is used when trimming a line to a desired length "n". D_1 identifies both the line and which end of the line is to be lengthened or shortened to a length of "n".
TRIM LIN LNG6 : ENT D_1)

C. Trimming Circles, Arcs, and Fillets

Circles are created in counterclockwise dirction beginning and ending at the 3 o'clock position. They are also trimmed in a counterclockwise direction.



Basic Trim Circle Command

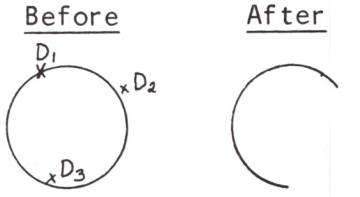
TRIM CIR : ENT D_1 DIG D_2 D_3)

D_1 Identifies the circle or arc to be trimmed and which end of the circle or arc is to be relocated at position D_2.
D_2 Gives the new location of the end to be trimmed.
D_3 The third digitize is optional. If present it will relocate the other end of the arc.

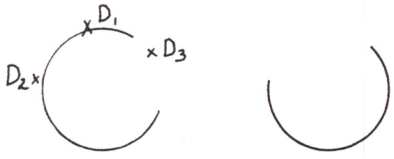
If the circle or arc to be trimmed is selected on the beginning half, the resulting arc will go counterclockwise from D_2 to D_3. If the arc or circle to be trimmed is selected on the ending half the resulting arc will go counterclockwise from D_3 to D_2.

Examples

TRIM CIR : ENT D_1 DIG D_2 D_3)

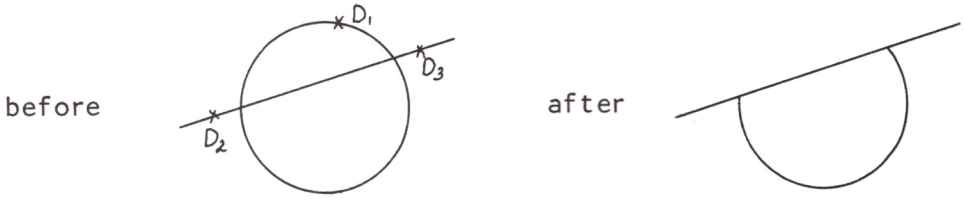


TRIM ARC : ENT D_1 DIG D_2 D_3)



Trimming a circle using the modifier INTOF.

TRIM CIR INTOF : ENT D_1 ENT D_2 D_3)



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Part II
Divide Entity Command

This Feature is Present on the Computervision CADDS3® System but has not been implemented on MATC CAD.

The DIV ENT command will break up an entity into two or more parts and will allow portions of an entity (line, circle, arc) to be changed to a different line font, or totally erased. The break locations are specified by dropping a perpendicular from the digitized location to the entity to be divided. If the modifier INTOF is used, the break location will occur where the intersecting entity crosses or would cross the entity to be divided. The break locations are always in pairs and the line font modifiers are effective from digitizes D_2 - D_3 , D_4 - D_5 , etc. The remaining portions retain the original entity line type.

Example: Dash font used.

DIV ENT DASH : ENT D_1 DIG D_2 D_3 D_4 D_5)

Before After

To accomplish this, the digitizes must follow the direction established by the first two digitizes. Any out-of-order digitize could produce unpredictable results.

Basic Divide Entity Command

DIV ENT (Modifiers) : ENT D DIG D D)

Divide Entity Modifiers

ERS The specified portions of the selected entity are erased. (default)

INTOF Allows designation of portions by selecting intersecting entities.

DASH Specified portions are changed to a "dash" line font.

SOLID Specified portions have a solid line font. If solid to begin with, the entity will look the same but will be made up of new smaller pieces.

No Modifier: The command goes to the default value of ERS.

The INTOF modifier can be used with any of the modifiers.

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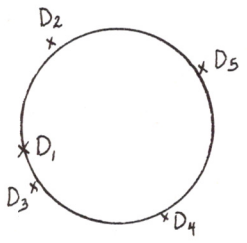
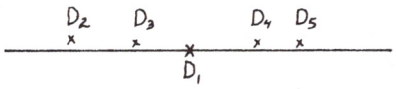
This Feature is Present on the Computervision CADDS3® System but has not yet been implemented on MATC CAD.

1. # DIV ENT: ENT D_1 DIG D_2 D_3 D_4 D_5)
   # RPNT)

Before 

After

2. DIV ENT : ENT D_1 DIG D_2 D_3 D_4 D_5)

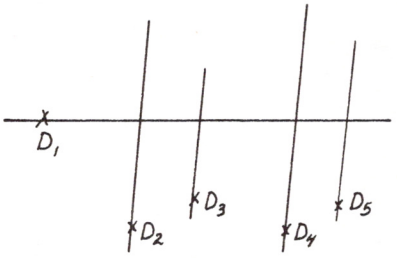
Before 

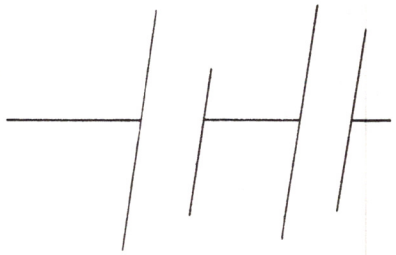
After _____ _____ _____

3. INTOF Modifier

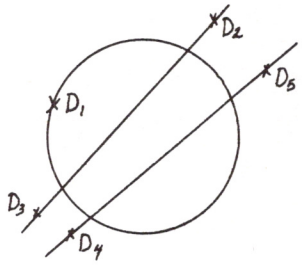
   # DIV ENT INTOF : ENT D_1 ENT D_2 D_3 D_4 D_5)
   # RPNT)

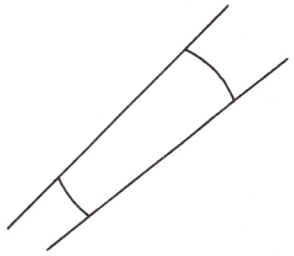
LINE

Example 



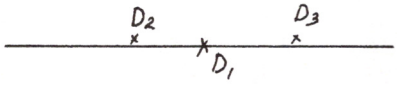
CIRCLE

Example 



4. Font Modifiers
   In this example the divided parts are given a dashed line font.

   # DIV ENT DASH : ENT D_1 DIG D_2 D_3)
   # RPNT)

Before 

After _____ - - - - - - - _____

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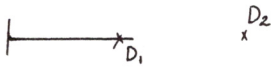
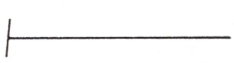
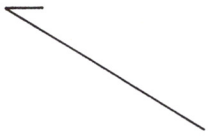
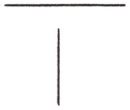
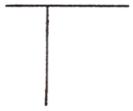
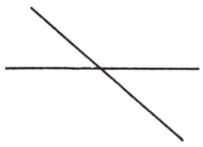
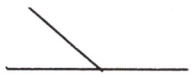
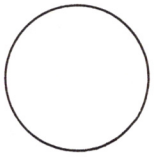
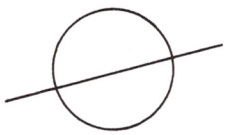
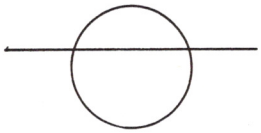
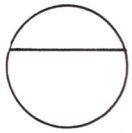
UNIT 11 EXERCISE SECTION

Part I
Trimming Lines and Circles

Directions: Write the command language as it would appear on the command display and give the required digitizes that would be used to make the following modifications to the following exercises.

BEFORE	AFTER	Command Language
		# TRIM LIN : ENT D_1 DIG D_2)
1.		1.
2.		2.
3. 		3.
4. 		4.
5. 		5.
6. 		6.
7. 		7.

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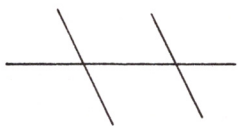
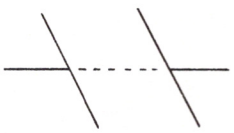
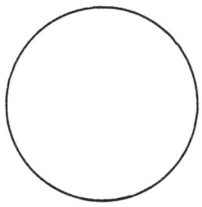
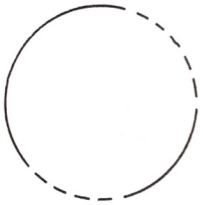
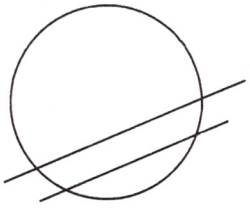
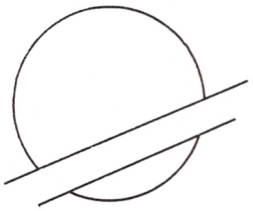
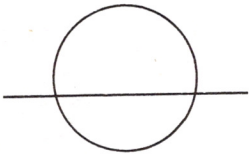
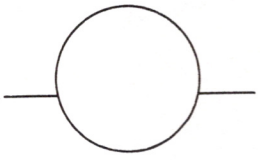
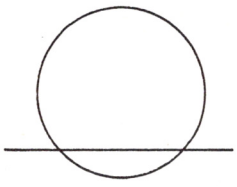
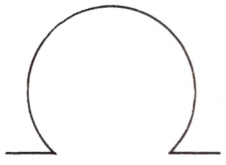
Part II
Dividing Entity Exercises

This Feature is Present on the Computervision CADDS3® System but has not yet been implemented on MATC CAD.

Directions: Write the command language as it would appear on the command display and give the digitizes necessary to alter the following entities as required.

	BEFORE	AFTER	Command Langauge
1.			1.
2.			2.
3.			3.
4.			4.
5.			5.
6.			6.

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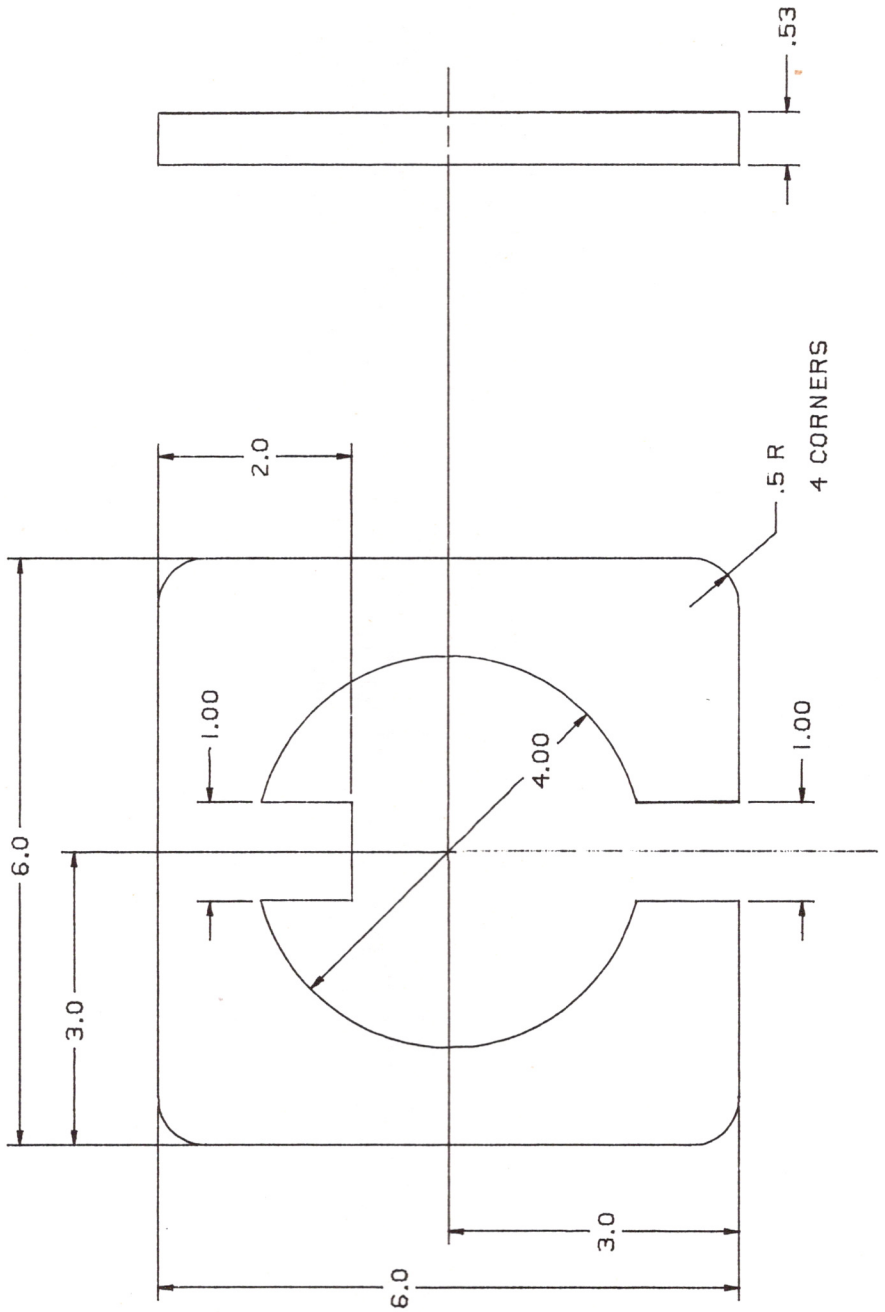
UNIT 9 LAB EXERCISE

Directions: Using the TRIM and DIV ENT commands as needed, construct and dimension the following drawings on the graphics design station. Use the part name _____ for Exercise 9A and the part name _____ for Exercise 9B. Upon completion have your instructor check them.



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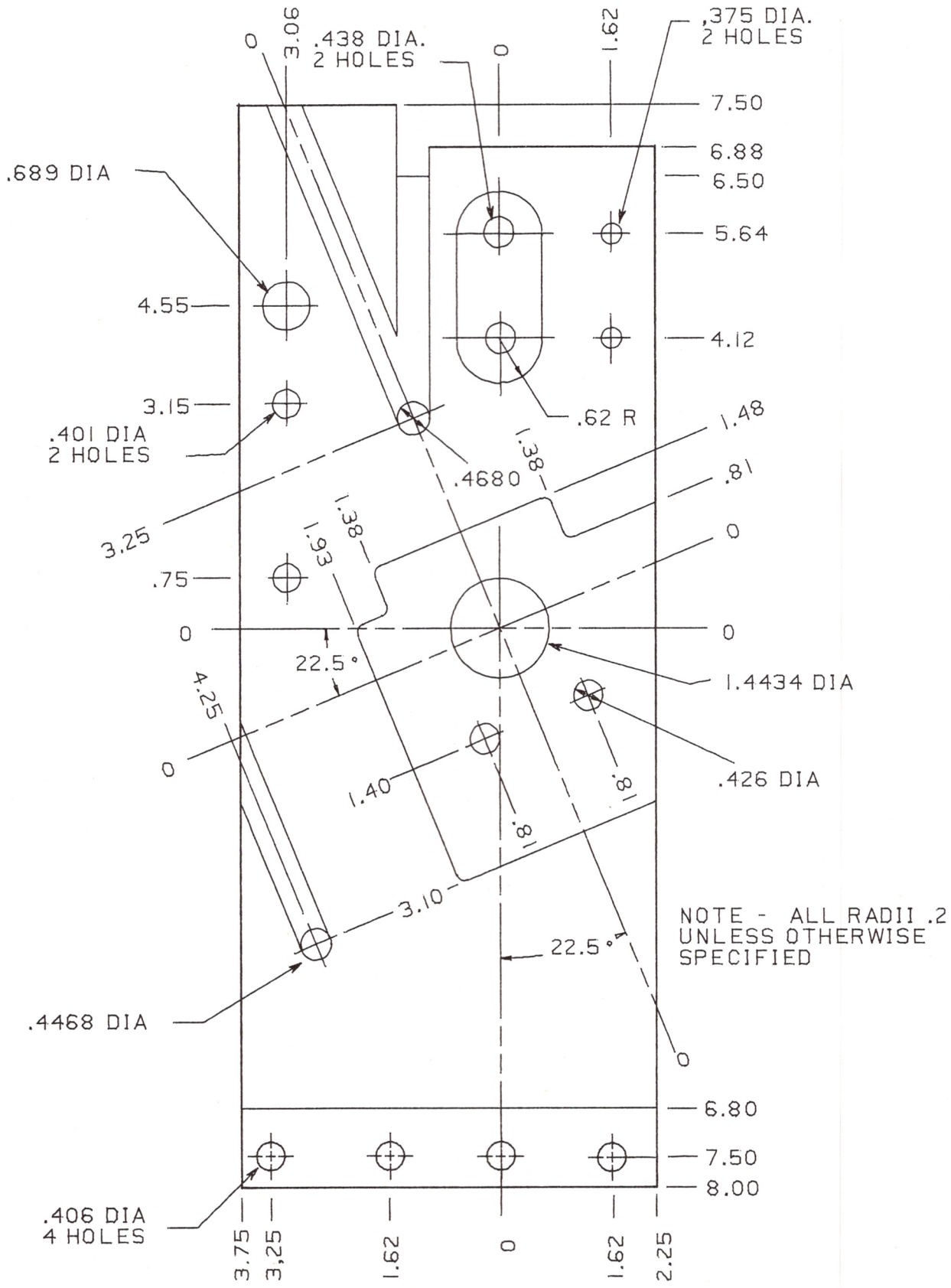
Exercise 9B



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COMPUTER-AIDED DRAFTING

Unit 10
Entity Verification and Measurement

MATCCAD™

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Introduction

A useful feature of a graphics system is the ability to verify that the lines, circles, etc., of your drawing have been correctly entered into the part database. Other features allow you to measure the distance or angle between two lines or the perimeter of an enclosed area. This unit will give you instruction and practice in verifying the size, locations, and attributes of entities, and in measuring the angle and distance between lines and/or points. The information obtained with these commands will give you important feedback in verifying the accuracy of your drawing and also finding the unknown value of various distances, lengths, angles, and positions in existing drawings.

Objectives

At the completion of this unit you will be able to:

1. Verify different entities of a drawing on the data display of a computer graphics system using the entity verification (VER) command.

2. Measure angles and distances between lines, and measure distances between locations using the measure (MEAS) command.

3. Answer specific questions about the size and location of entities and angles of a drawing on the data display of a computer graphics system by using the verify (VER) and measure (MEAS) commands.

4. Use the On-Line Documentation feature as a ready reference to the nouns, verbs, and modifiers available on the system.

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Part I
The Verify Command

The VERIFY (VER) command will verify entities such as lines, circles, points, etc., for size and location by printing the data on the command display. The VERIFY command allows the operator the option of verifying all the entities on the screen, or entities can be individually selected for verification by digitizing.

A. Command For Verifying All Entities

VER ↵ Will verify all entities on the screen.
To stop the screen listing enter CTRL-S
To restart verification listing enter a second CTRL-S

B. Command For Verifying Specific Entities

VER : ENT D D . . . ↵ Verifies each entity selected.

Each entity selected by the digitizing pen is verified immediately.

Following the ENT prompt the user can insert a LIN, CIR, ARC, PNT, etc., mask so that entity selection is limited to the entity type desired.

When a line is verified the listing will show the layer of the line, the line font type of the line (1=SOLID, 2=DASH), the XY coordinates of both ends, and the true length.

When an arc or circle is verified the screen will list out the layer of the arc or circle, the line font type (1=SOLID, 2=DASH), the XY coordinate of the origin, the radius, the beginning angle in degrees, and the angular extent of the arc.

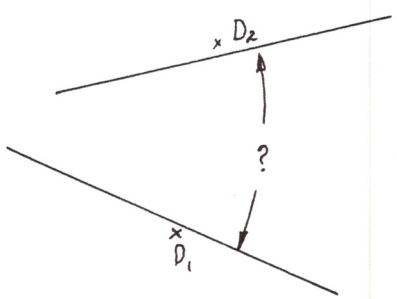
Part II
The Measure Command

A. Measuring Angles

The MEASURE ANGLE (MEAS ANG) command measures the angle between two lines or the angle of a line from the horizontal axis. The acute and obtuse angle values are both given.

MEAS ANG : ENT D_1 D_2 ENT . . . ↵

The acute and obtuse angles are displayed. The unit returns to ENT for the next angle selection.



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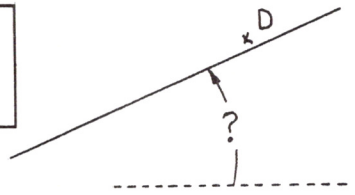
#MEAS ANG HOR : ENT D)

This Feature is Present on the Computervision CADDS3® System but has not yet been implemented on MATC CAD.



The acute and obtuse angles of the line to the horizontal are given.

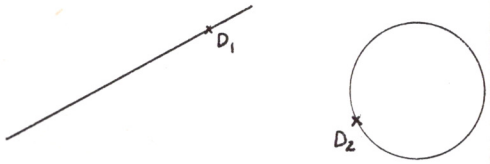
When you are ready to leave the # MEAS ANG command, press the RETURN key.

B. Measuring Distance

The MEASURE DISTANCE (MEAS DIST) command measures the distance between two locations. Typically you would type in a END, PNT or ORG mask to specify locations on existing entities.

Basic Command

MEAS DIST : DIG END D_1 ORG D_2)



Part III
On-Line Documentation

On the MATC CAD system a list of all available commands, all the nouns that go with a particular verb, or the modifiers for a specific verb-noun combination can be listed out using the on-line documentation feature.

A. To obtain a list of verbs enter a ?) following the CADDS prompt.

```
# ? )
 INS   ERS   TRI   TRM   MOV   ECH   EXI         LST   LIS
 ROT   FIL   REG   RPN   REP   MEA   PAG   COP   MIR   PRI
 RST   RES   ZOM   ZOO   CHG   PRN   PAR   PLO   PLT
 CHA   CHN   SEL   VER   CON         MEM
 SAV   QPN   DIM   DIS   DIV
```

B. To obtain a list of nouns for a particular verb, enter the selected verb (in this example INS for insert) followed by ?)

```
# INS ? )
  LIN   CIR   PNT   ARC   TXT
  TEX   FIL   LDI   RDI   ADI
  DDI   XH    SPL   BSP
```

C. To obtain a list of modifiers used with a selected verb-noun combination, enter the verb and noun followed by ?)

```
# INS LIN ? )
  ANG   FON   LNG   PRL   HOR
  PRP   TAN   VER
```

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UNIT 10 EXERCISE SECTION

Part I
Verification Exercise

Directions: Answer the following questions:

1. To verify all existing entities in a drawing, what command would be entered on the keyboard?

   _____

2. How would the operator temporarily stop the verification listing once it has started?

   _____

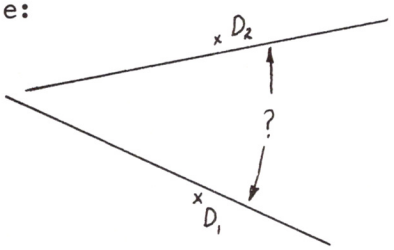
4. If you want to verify a specific circle on a drawing, what command will verify only that circle?

   _____

Part II
Measurement Exercises

Directions: Write the command language as it would appear on the telewriter to measure the following:

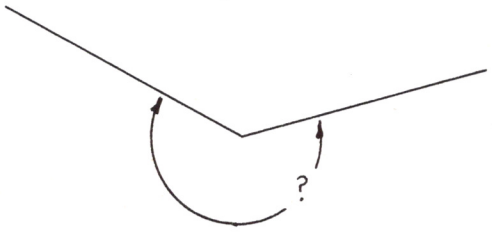
example:



Command Language

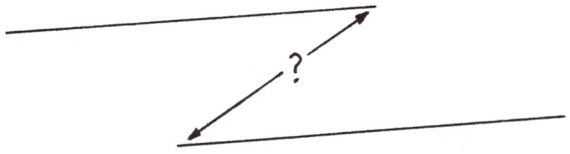
MEAS ANG : ENT D_1 D_2)

1.



1.

2.



2.

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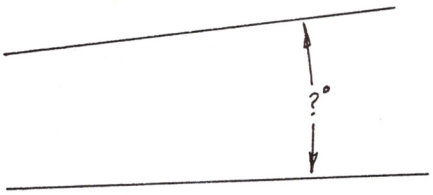
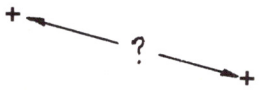
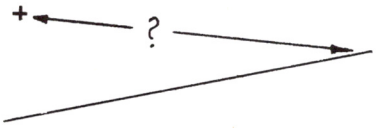
3. 

3.

4. 

4.

5. 

5.

6.

6.

Where are these points located?

7.

7.

Where are the centers of the circles?

Part III
On-Line Documentation Exercises

Directions: Answer the following questions about On-Line Documentation.

1. When a question mark (?) is entered after the CADDS prompt (#), what happens?

_____

2. If INS LIN ? is entered, what happens?

_____

3. How can you get a list of all nouns used with INS?

_____

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UNIT 10 LAB EXERCISE SECTION

Lab Exercise 10
On-Line Documentation

1. LOG IN the system.

2. Verify your answers from exercise _____ using the measure angle, measure distance, and entity verification commands.

3. Explore On-Line Documentation by answering the following questions:

   a. Is PAG a verb? _____

   b. What nouns can be used with the verb SEL?

   _____

   _____

   c. What modifiers can be used with the command INS TXT?

   _____

   _____

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Lab Exercise 10B

Directions: Recall Exercise 7B, part name _____ from the files. Answer the following questions about it using the verify (VER) and measure (MEAS) commands.

1. What are the X,Y coordinates of the upper end of line (12) ? _____
2. What is the length of line (12) ? _____
3. What is the angle between lines (1) and (12) ? _____
4. Where is the X,Y origin of circle (11) ? _____
5. What is the radius of circle (11) ? _____
6. What is the length of line (4) ? _____
7. What is the angle between lines (6) and (7) ? _____
8. Where would the point of intersection between lines (8) and (12) be?

   X = _____ Y = _____

9. Where is the X,Y origin of circle (5) ? _____
10. What is the X,Y coordinate at the point where line (4) is tangent to circle (5) ? _____
11. What is the angle of line (12) with respect to the horizontal ? (The positive X-axis is 0 degrees.)
12. What is the minimum distance between circles (1) and (2) ? _____
13. What is the perpendicular distance from the lower left endpoint of line (8) to line (10) ? _____
14. What is the distance from the center of circle (1) to the upper end of line (9) ? _____

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COMPUTER-AIDED DRAFTING

Unit 11
Inserting Dimensions and Text on Drawings

MATCCAD™

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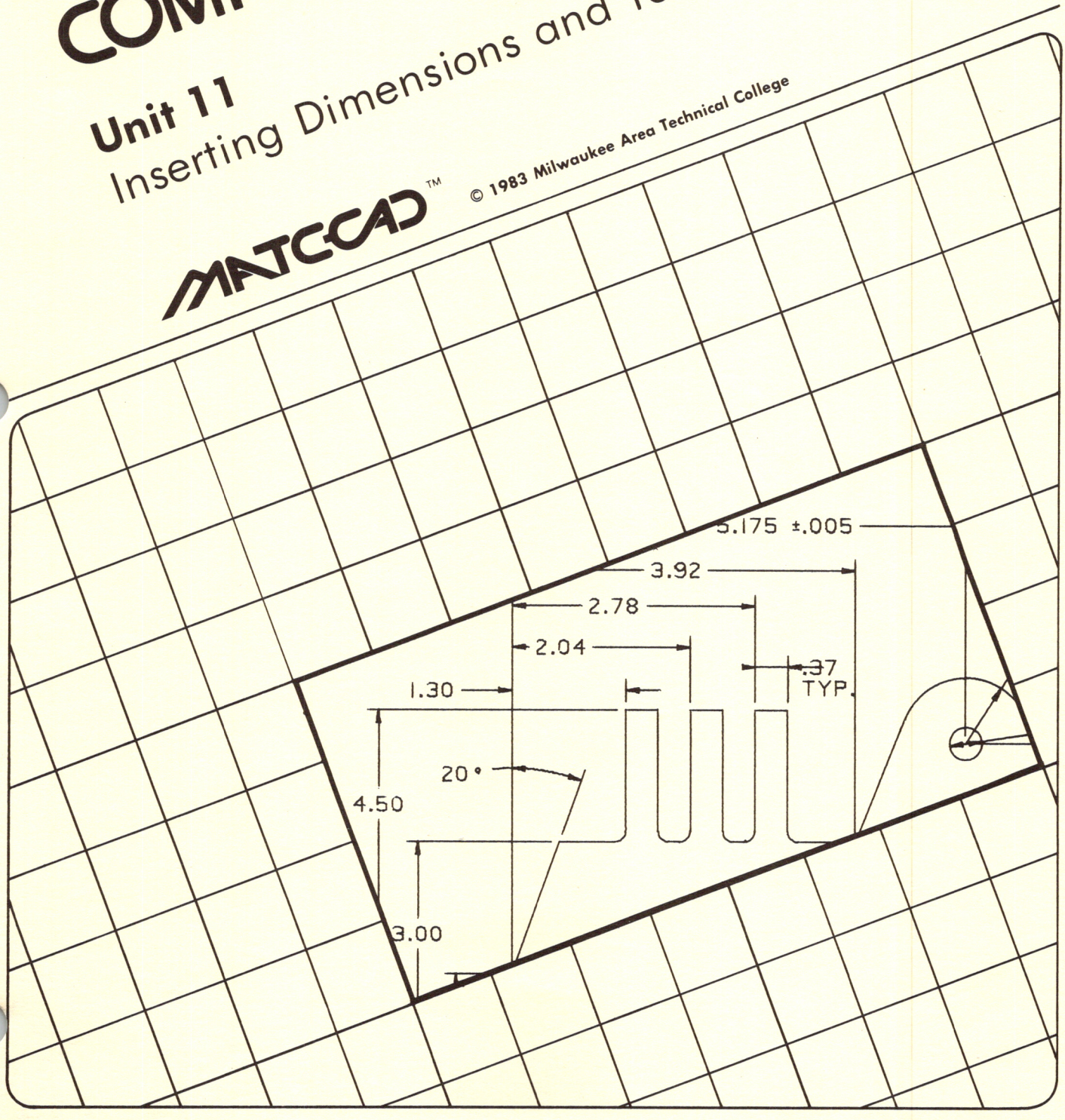

Introduction

This unit will give you practice in one aspect of documentation of drawings through the use of inserting linear, radial, diameter, and angular dimensions. You will also practice inserting text on drawings. This information will be used to construct, dimension, and insert text in a two-dimensional drawing using the computer graphics system. It will also be used to add dimensions and text to previously constructed parts.

Objectives

At the completion of this unit you will be able:

1. To write the commands needed to insert various dimension measurements on a drawing.

2. To write the commands needed to insert text and notes on a drawing.

3. To write the text and dimension modification commands needed to change the height, width, slant, location, etc., of existing text and dimensions.

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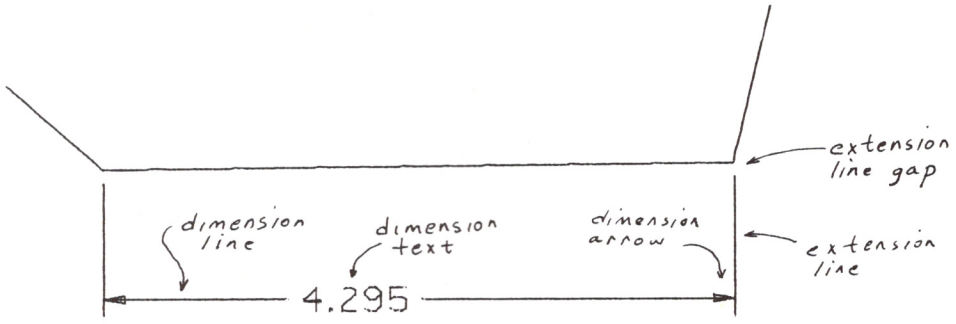
Unit 11 Reference Section
Dimensioning

Dimensions are used to indicate the necessary measurements of a part in a standardized way so that a person needing to interpret the drawing will understand how the part represented by the drawing is to be made.

A dimension created on a computer graphics system is made up of a number of automatically generated lines and text as shown in the following drawing:



Once the entities to be dimensioned and the location of the dimensioning text are specified, the dimension lines, extension lines, witness lines, arrowheads, and dimension text are all automatically calculated and displayed.

On our graphics system four types of dimensioning are supported.

1. Linear Dimensions
2. Radial Dimensions
3. Diameter Dimensions
4. Angular Dimensions

Remember that the drawscale option (#SEL PLOT DSCLn) can be used to enlarge or reduce the displayed (and plotted) size of the dimension text, arrowheads, and extension line gap. Refer to chapter 5 page 5-3 for details.

Dimensioning Modifiers

There are several modifiers which are common to all four types of dimensioning.

General Dimensioning Modifiers

PRECn — Precision determines number of digits following the decimal point for the automatically generated dimensioning text. Default is 3 digits. The value of n can be from 0 to 5 digits.

HGTn — Used to specify the text height. Default is .156"

TXT ' ' — Allows entering alternate text or numbers in place of the automatically generated dimension text.

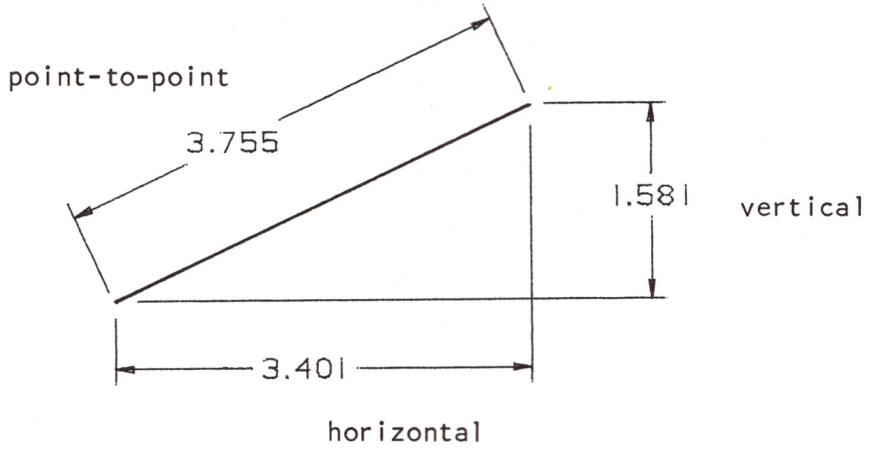
ATXT ' ' — Allows adding additional text following the dimension text value. A space must follow the TXT or ATXT modifiers.

This Feature is Present on the Computervision CADDS3® System but has not yet been implemented on MATC CAD.

Part I
Linear Dimensioning

A linear dimension is inserted in a drawing to indicate the distance between two locations. These locations can be the ends of entities, points, or the intersection of entities. Two digitizes are required to select the ends of two entities to be dimensioned. A third digitize is needed to indicate the location of the dimensioning text.

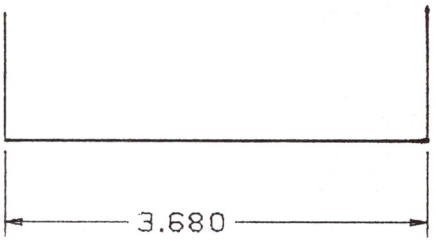
There are three types of linear dimensions: horizontal, vertical, and point-to-point. A horizontal dimension will indicate the horizontal distance between two locations. A vertical dimension will indicate the vertical distance between two locations. A point-to-point dimension will indicate the true distance between two locations.



Basic Linear Dimensioning Command

INS LDIM (modifiers) : ENT D_1 D_2 DIG D_3)

The digitizes D_1 and D_2 indicate the ends of lines or points being dimensioned. The third digitize D_3 is required to specify the location of the lower left corner of the dimensioning text.



Linear Dimensioning Modifiers in Addition to PREC, HGT, TXT

HOR	Indicates that the horizontal dimension between 2 entity locations is to be generated.(This is the default condition.)
VER	Used when inserting vertical dimensions.
PPNT	For creating point-to-point dimensions. The dimension line will be parallel to a straight line joining the entity locations to be dimensioned. The generated dimension text indicates the true distance between the two locations.

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SUPn — Used for suppressing extension lines. When n equals:
0 - none suppressed (default)
1 - suppress the first extension line
2 - suppress the second extension line
3 - suppress both extension lines
4 - suppress the first dimension and extension lines
5 - suppress the second dimension and extension lines
6 - suppress both dimension and extension lines

AO — Place the arrows outside of the extension lines.

AI — Place the arrows inside of the extension lines. (default)

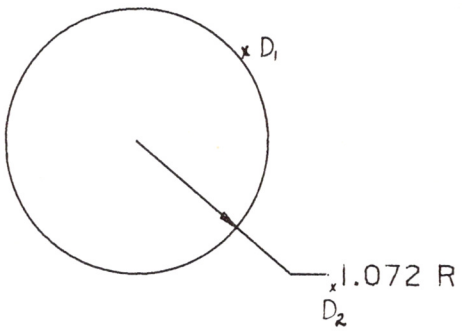
Part II
Radial Dimensioning

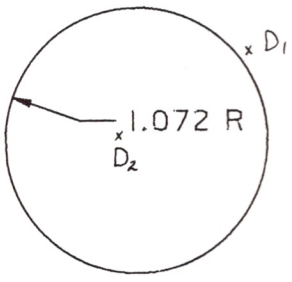
The radial dimension command is used for generating a dimension for the radius of circles or arcs. It requires one entity selection for identification of the arc to be dimensioned, and one digitize for the placement of the dimensioning text.

Basic Radial Dimensioning Command

INS RDIM (modifiers) : ENT D_1 DIG D_2)

D_1 selects the entity to be dimensioned
D_2 specifies the location of the lower left corner of the dimension text. The leader line for the dimension will go to the head (front) of the text.

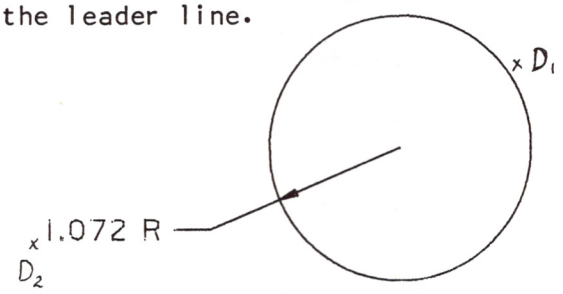




Radial Dimensioning Modifier in Addition to PREC, HGT, TXT

This Feature is Present on the Computervision CADDS3® System but has not yet been implemented on MATC CAD.

END — Will draw a tail from the end of the dimension text to the leader line.



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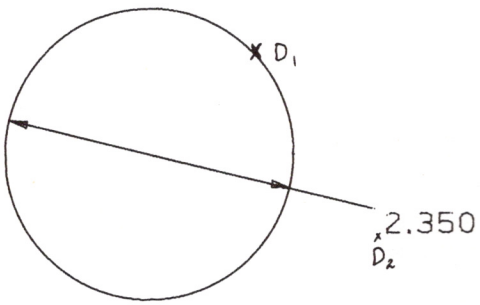
Part III
Diameter Dimensioning

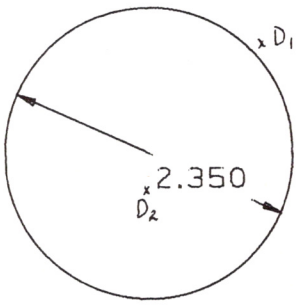
The diameter dimensioning command is used to generate a dimension for diameter of a circle.

Basic Diameter Dimensioning Command

INS DDIM (modifiers) : ENT D_1 DIG D_2)

D_1 specifies the circle to be dimensioned
D_2 specifies the location of the dimension text





Part IV
Angular Dimensioning

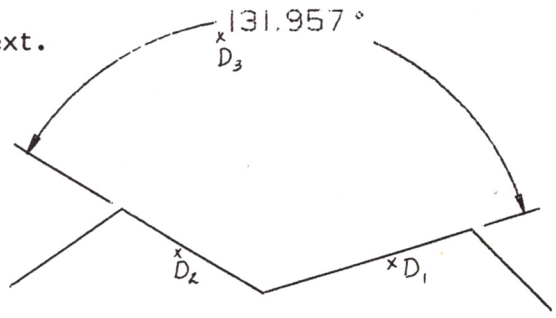
The angular dimensioning command is used to show the angle between two lines.

Basic Angular Dimensioning Command

INS ADIM (modifiers) : ENT D_1 ENT D_2 DIG D_3)

The lines used for the the angular dimension must be selected in a counterclockwise direction.

D_1 specifies first line to dimension to
D_2 specifies second line
D_3 specifies the location of the dimension text.



If a dimension line arrow falls on the line to be dimensioned the extension line on that side will be suppressed automatically.

Angular Dimensioning Modifier in Addition to PREC, HGT, TXT

This Feature is Present on the Computervision CADDS3® System but has not yet been implemented on MATC CAD.

AO Places the arrows outside of the extension lines. (Text In Arrows Out) This is slightly different than Computervision.)

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Part V
Changing Dimensions

The change dimension command modifies the characteristics of an existing dimension.

Basic Change Dimension Command

CHG DIM (modifiers) : ENT D)

Change Dimension Modifiers

ORG — Change the dimension location by digitizing the origin of an existing dimension and then the desired new location. The origin of a dimension is at the lower left corner of the dimension text.

TXT — Replace the existing dimension text with different text. You must use the single quote delimiter ' when specifying new text.

SUPn — Change the supression status of the extension and dimension lines.

0 - all lines present
1 - first line suppressed
2 - second line suppressed
3 - both lines suppressed
4 - suppress the first dimension and extension
5 - suppress the second dimension and extension lines
6 - suppress both dimension and extension lines

HGTn — Change the text height of the specified dimension to the value 'n'.

AI — Change arrows to inside the extension lines.

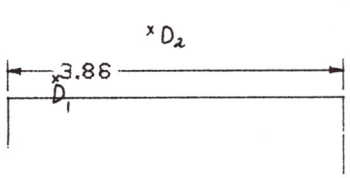
AO — Change arrows to outside the extension lines.

This Feature is Present on the Computervision CADDS3® System but has not yet been implemented on MATC CAD.

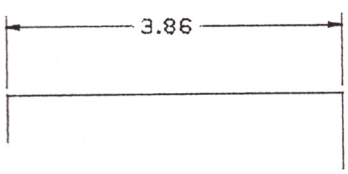
Examples

CHG DIM ORG : ENT D_1 DIG D_2)

before



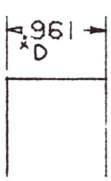
after



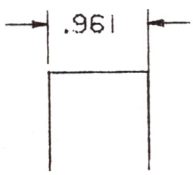
This Feature is Present on the Computervision CADDS3® System but has not yet been implemented on MATC CAD.

CHG DIM AO : ENT D)

before



after



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Part VII
Text Commands

A text entity is made up of a string of text characters, the location of the text, and the various text parameters which affect the orientation and appearance of the text string when it is displayed.

A. Insert Text Command

The **Insert Text** command is used to specify text and its location.

Insert Text Basic Command

```
# INS TXT /TEXT GOES HERE/ : DIG D )
```

D* TEXT GOES HERE

Delimiters: Each string of text is delimited at the beginning and end by a pair of either ' " or / characters.

Insert Text Modifiers

HGTn	Specify text height. (Default is .156)
WDTn	To specify the text width (The default is 80% of the height)
SLTn	To specify the slant of the text characters from -45 to 45 degrees.
ANGn	To specify the base angle of the line of text. Horizontal is 0 degrees.
LJT	Left justified text. (Default).
CJT	Center justified text.
RJT	Right justified text.

Examples

```
# INS TXT "MDT-108 SECT. 1 )
  EX. NO. 4 )
  R. SMITH )
  B. JONES )
  D. ANDREWS )
  DEC. 25, 1982" HGT.5 CJT : DIG D )
```

This text is one note (string) on 6 lines and it is center justified. A carriage return follows each line.

D

MDT-108 SECT. 1
EX. NO. 4
R. SMITH
B. JONES
D. ANDREWS
DEC. 25, 1982

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B. Select Text Command

The **Select Text** command allows the user to change the default text parameter values for height, width, slant, etc. Any text inserted in your drawing at a later time will take on the characteristics you have chosen with the select text command.

Select Text Basic Command

SEL TXT (modifiers))

Select Text Modifiers

HGTn	To specify height of later inserted text. (Default is .156)
WDTn	To specify the text width. Default is 80% of the specifiec height.
SLTn	To specify the slant of the individual text characters from -45 to 45 degrees.
ANGn	To specify the base angle of the line of text.
CJT RJT LJT	To specify whether the text to be inserted will be center, right or left justified.

C. Change Text Command

The **Change Text** command allows changing selected characteristics of existing text.

Change Text Basic Command

CHG TXT (modifiers) : ENT D)

RPNT) the RPNT will clean up the displayed image and show the resultant changes

Change Text Modifiers

HGTn	Change the height to a selected size, "n".
ANGn	Change the base angle of the text line.
SLTn	Change the slant of the letters from -45 to 45 degrees.
WDTn	Change the character width.
LJT CJT RJT	Change to left, center, or right justified text.

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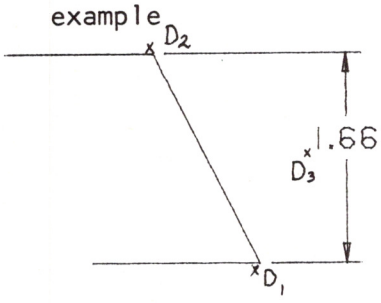
UNIT 11 EXERCISE SECTION

Part I
Dimensioning Exercise

Directions: Write the command language as it would appear on the command display and give the necessary digitizing locations needed for dimensioning the following exercises.

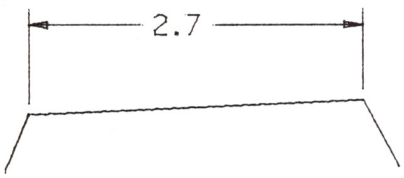
example



Command Language

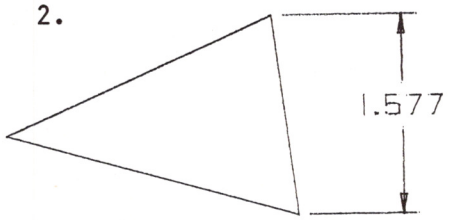
INS LDIM VER PREC2 :ENT D_1 D_2 DIG D_3)

1.



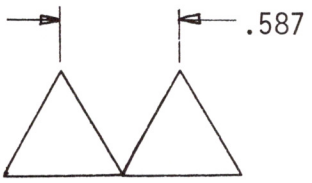
1.

2.



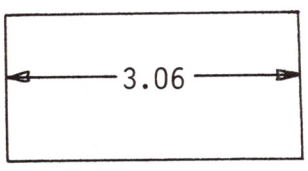
2.

3.



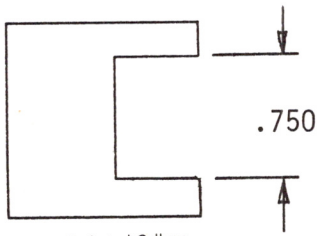
3.

4.



4.

5.



5.

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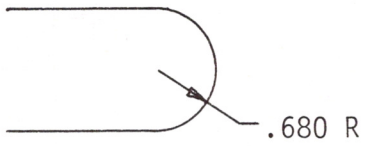
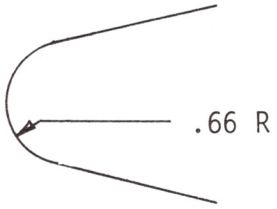
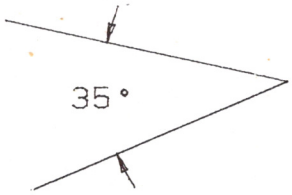
6. 

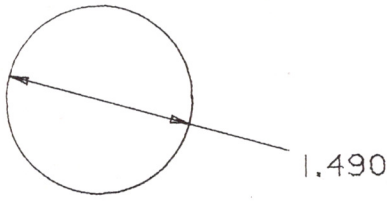
6.

7. 

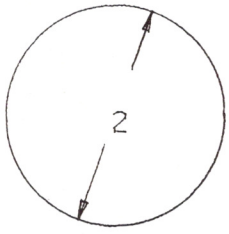
7.

8. 

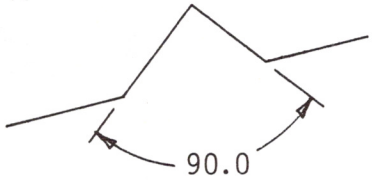
8.

9. 

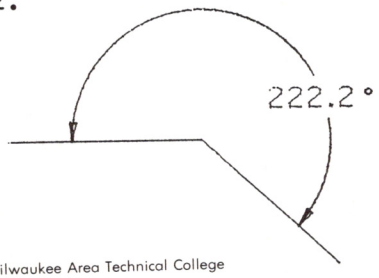
9.

10. 

10.

11. 

11.

12. 

12.

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Part II
Text Exercises

Directions: Write the command language as it would appear on the command display and show the location digitizes needed to insert the following text.

example:

ALL FILLETS .25R

D

Command Language

INS TXT /ALL FILLETS .25R/ : DIG D)

1. FAO

1.

2. ALL UNSPECIFIED
   TOLERANCES + .012

2.

3. 36" (1 CU. YD.)
   BUCKET

3.

4. R. DOE
   E. EDWARDS
   D. JOHNS

4.

Directions: Write the language as it would appear on the command display and show the required digitizes needed to change the following.

example:

D EXERCISE 14

EXERCISE 14

Command Language

CHG TXT HGT.30 : ENT D)
RPNT)

1.

WET GEARBOX

WET GEARBOX

1.

2.

3 HOLES

3 HOLES

2.

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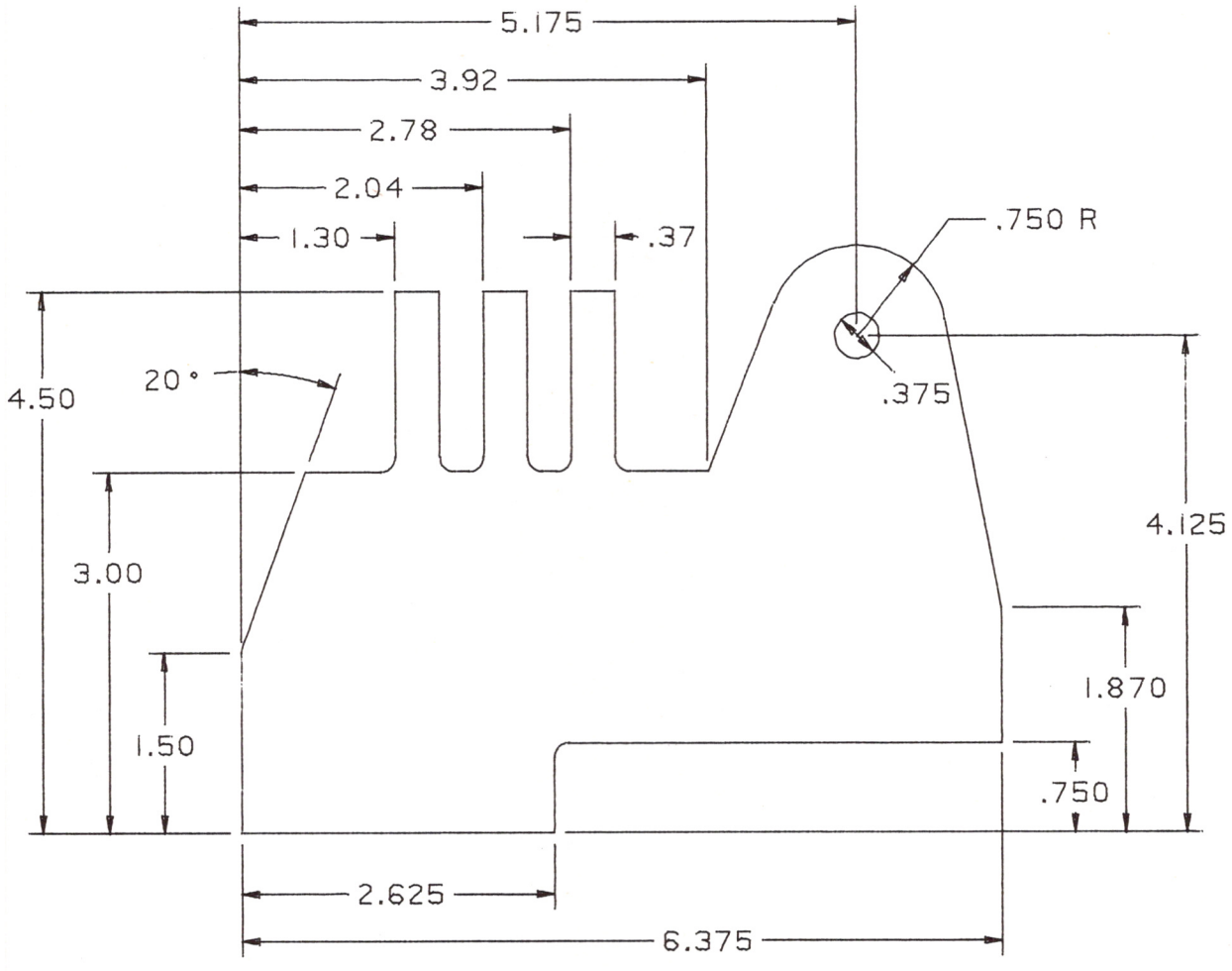
Unit 11 Lab Exercises
Drawing Exercise

Directions: Construct, dimension, and insert the required text for the following drawing on the graphics design station. Use the part name _____ for exercise 11. Upon completion, have your instructor check it.

Directions: Recall exercise _____ and exercise _____ from the system's files. Dimension and insert the required text for them using the computer graphics system. Upon completion, have your instructor check them.



ALL FILLETS 0.125 RADIUS

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COMPUTER-AIDED DRAFTING

Unit 12
Mirror, Move, and Rotate Commands

MATCCAD™

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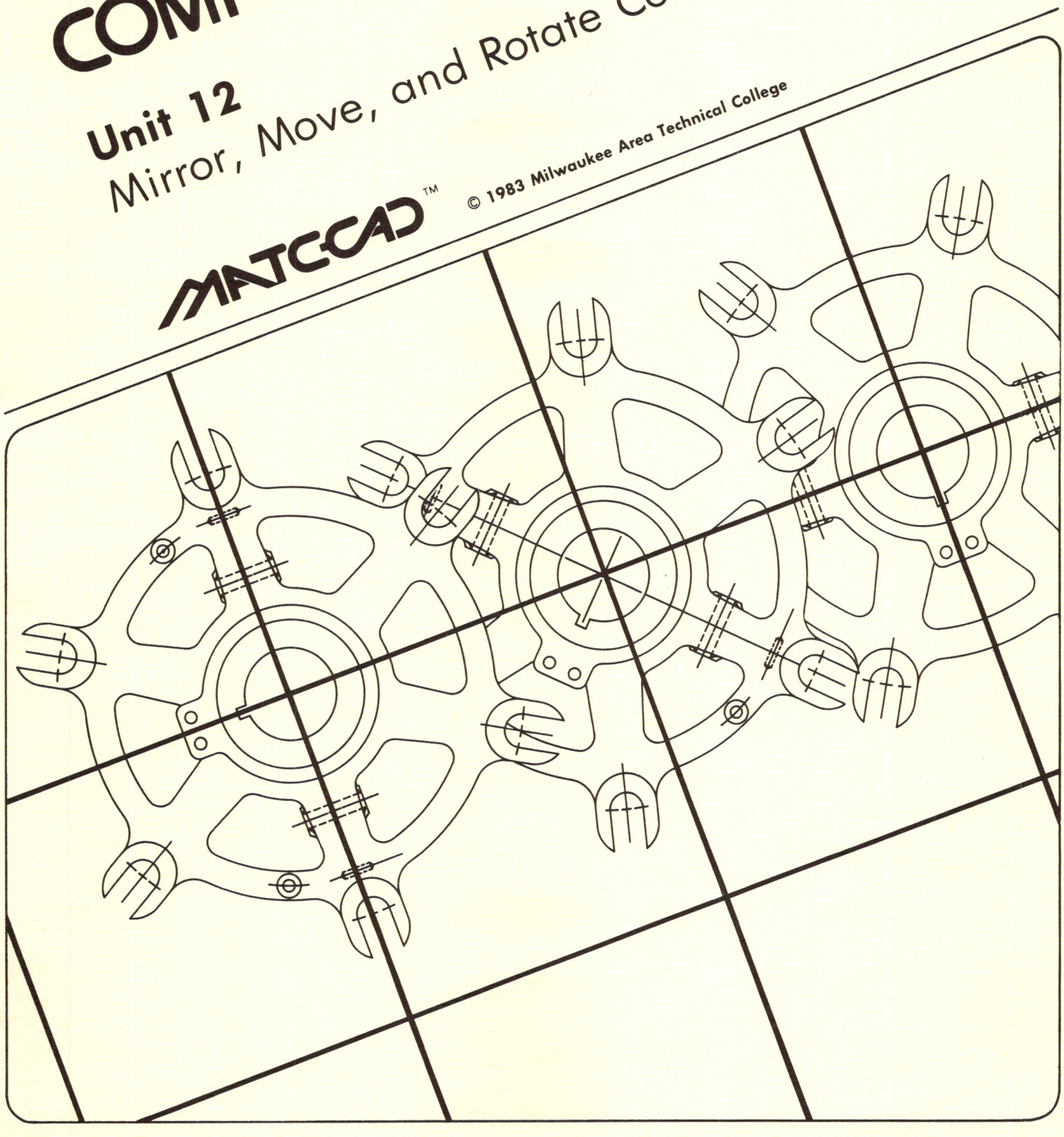

Introduction

A certain amount of work in computer graphics drawing construction involves parts that are symmetric about a given axis or have multiple repetitions of a particular feature equally spaced along a line, or at equally spaced angles a constant distance about some center. This unit will give instruction and practice in the use of the mirroring, translation (moving), rotation, and copying commands as an aid in the construction and modification of two-dimensional drawings with axial or radial symmetry or repetitive features.

Objectives

At the completion of this unit you will be able to:

1. Use the mirroring (MIR), rotation (ROT), and translation (MOV or TRANS) commands along with the COPY modifier to construct and modify entities and patterns of entities on a computer graphics system.

2. Construct, dimension, and modify assigned drawings on the computer graphics system by using the MIR, MOV, and ROT commands.

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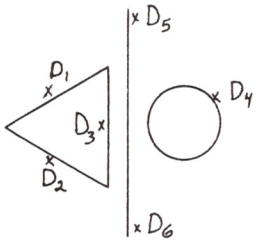
Part I
The Mirror Command

The mirror (MIR) command allows the creation of a mirror image of selected entities about a specified axis. The axis of mirroring can be the ends of a line, free digitizes, entered coordinate values, or any combination of these.

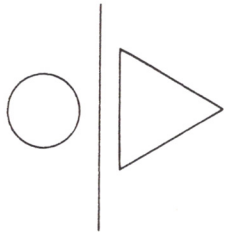
The entities to be mirrored are selected one by one, with a comma indicating the completion of entity selection. The DIG prompt following the comma indicates the user is to then enter two locations representing the axis of mirroring. These locations can be specified by either digitizing, coordinates, END or PNT masks.

MIR : ENT D_1 D_2 D_3 D_4 , DIG END D_5 D_6)

Before



After



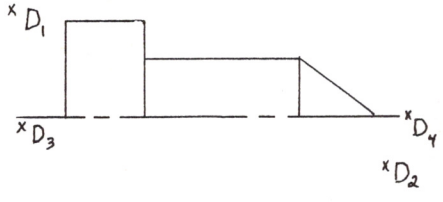
Mirror Modifier

COPY is the only modifier available with the MIR command. Using MIR COPY, the entities selected to be mirrored are symmetrically copied on the other side of the mirroring axis.

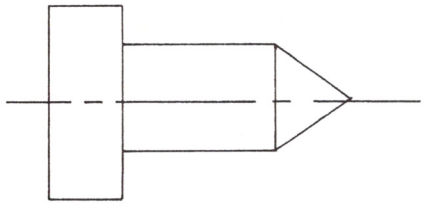
In the following example a WIN mask is used to select entities and the END mask is used for the selection of the mirroring axis.

MIR COPY : ENT WIN D_1 D_2 , DIG END D_3 D_4)

Before



After



Part II
The Move (Translate) Command

The move (MOV) or translate (TRAN) command will move existing entities from their current positions to a different XY location. The entities to be moved are selected one by one, with a comma indicating the completion of entity selection. The DIG prompt following the comma indicates the user is to then enter two locations by digitizing, coordinates or END masks. The direction and distance of movement is determined by the direction and distance between the two locations specified after the comma. Using the COPY modifier, various copies of the entities selected can be moved to other locations.

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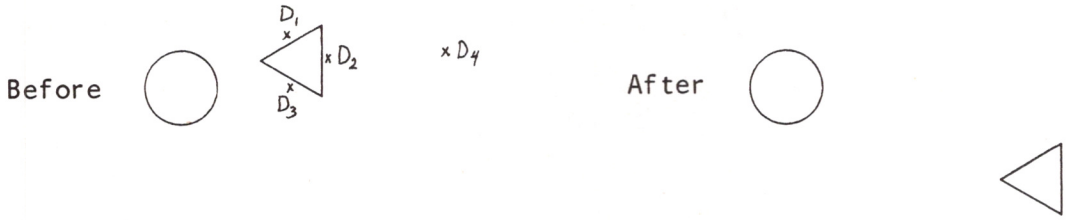
MOV : ENT D_1 , DIG $D_2 D_3$)

Specifying the MOV Direction and Distance with Coordinates

In this example the selected entities are moved 4 inches left and 3 inches down from their present position.

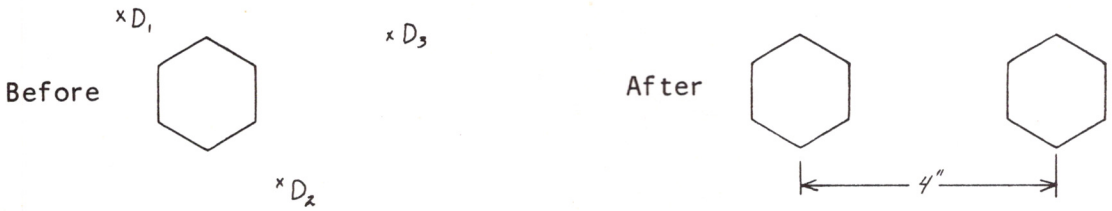
MOV : ENT $D_1 D_2 D_3$, DIG D_4 IX4IY-3) D_4 can be anywhere on the screen.



Move Modifiers

COPY - The COPY modifier with the MOV command is used to move a copy of existing selected entities to a new location while the original entities remain in place.

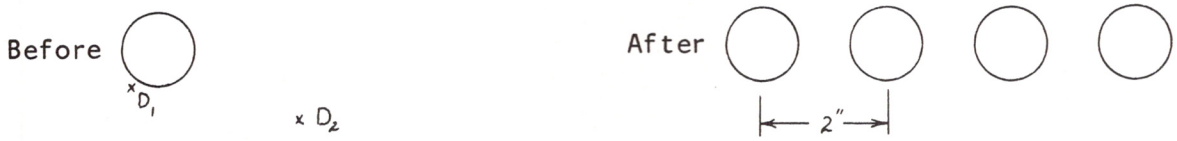
MOV COPY : ENT WIN D_1 D_2 , DIG D_3 IX4) D_3 can be anywhere on the screen



For multiple copies, the first location specification in the command after the comma is followed by some incremental coordinate value followed by Nn where n is the number of copies that will be made.

Example: To make 3 copies spaced 2" apart, you would enter IX2 followed by N3.

MOV COPY : ENT D_1 , DIG D_2 IX2N3) D_2 can be anywhere on the screen



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Part III
The Rotate Command

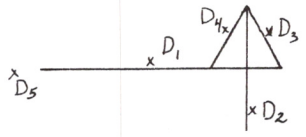
The rotate (ROT) command is used to rotate selected existing entities about a specified center of rotation. The major application of the command is to create rotated copies of an entity or entities at equally spaced angles at a constant distance from some center of rotation.

The entities to be rotated are selected one by one with a comma indicating the completion of entity selection. The DIG prompt following the comma indicates that the user is to then specify a center of rotation. This location is usually specified by an XY coordinate or by referencing the END or ORG of an existing entity. Depending on the form of the command, the center of rotation can be followed by digitizing or incremental coordinates.

In this example the selected entities are rotated 90 degrees to a new position.

ROT ENT ANGn : ENT D_1 . . D_4 , DIG D_5)

Before



After

Rotation Modifiers

COPY The selected entities are rotated and copied. (The default is no copying.)

ANGn The entities selected are rotated by a specified angle "n" in a counterclockwise direction about the specified center of rotation.

There are three basic ways to use the rotate command with its associated modifiers:

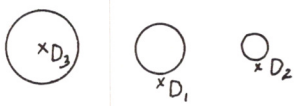
1. Rotating by a specified angle
2. Rotating by an unspecified digitized angle
3. Rotating multiple copies by a specified angle

1. Rotating a Copy a Specified Angle

In this example the entities selected are rotated 60 degrees and copied.

#ROT COPY ANG60 : ENT D_1 D_2, DIG D_3)

Before



After

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2. Rotatating by an Unspecified Digitized Angle

In this example the angle of rotation is determined by entering three locations following the comma and DIG prompt.

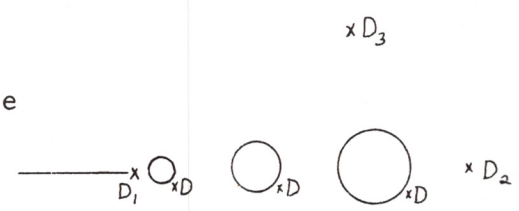
D_1 indicates the center of rotation
D_2 indicates a reference mark for the beginning of the rotational angle
D_3 specifies the finish of the rotational angle

The angle that each entity is rotated is determined by the angle between the lines D_2 D_1 and D_3 D_1 .

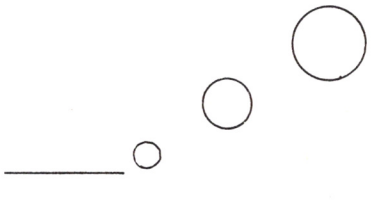
Typically, D_1 D_2 D_3 will be entered as coordinate values, or as the END or ORG of existing entities.

ROT : ENT D . . D, DIG D_1 D_2 D_3)

Before



After



3. Rotating Multiple Copies by a Specified Angle

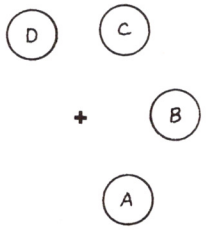
In this case the rotational angle is specified as a numeric value following the ANG modifier. The multiple copies are obtained by adding Nn, where "n" indicates the number of additional copies (in addition to the first copy) that the user wants to make.

In the following example, the original circle was rotated and copied about an existing point three times at 60 degree increments.

ROT COPY ANG60 : ENT D_1 , DIG PNT D_2 N2)

Before

After



Note that we end up with four circles. A is the original entity, B is the original copy, and C and D are the result of adding N2 following the selection of the existing point as the center of rotation.

This command to rotate multiple copies is slightly different than the Computervision system. On the Computervision system, for the above example, you would have to enter the incremental coordinate IZ0 just before entering N2.

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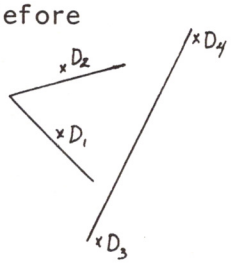
Unit 12 Exercise Section

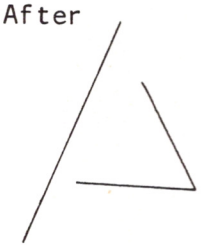
Part I
Mirroring Exercises

Directions: Write the command language and show the required digitizes that would be needed to modify or construct the following using the MIR command.

Before



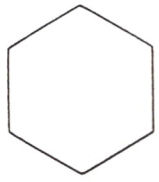
After



Command Language

MIR : ENT D_1 D_2 , DIG D_3 D_4)

1.

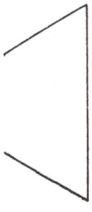


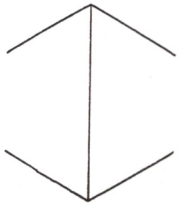
1.

2.

2.

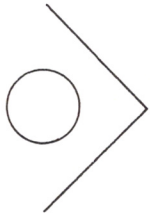
3.

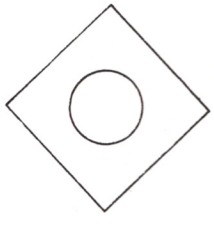




3.

4.





4.

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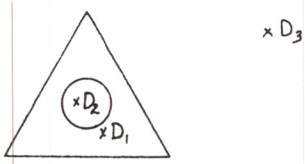
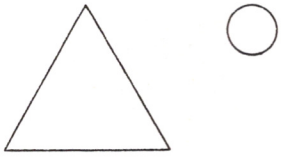
Part II
Moving (Translation) Exercises

Directions: Write the command and show the required digitizes that would be needed to modify or construct the following examples using the MOV command.

Before



After

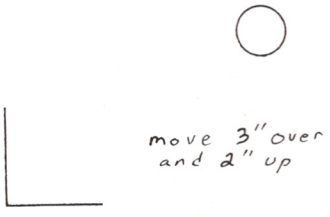


Command Language

MOV :ENT D_1, DIG D_2 D_3)
or

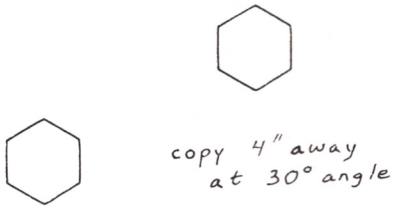
TRANS : ENT D_1, DIG D_2 D_3)

1.



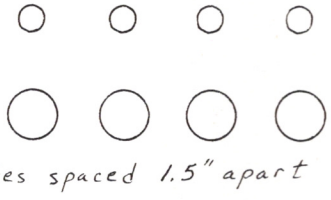
1.

2.



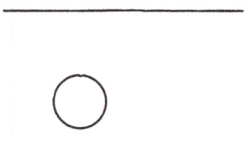
2.

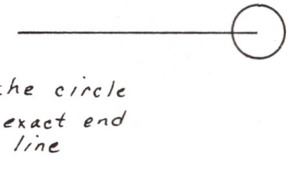
3.



3.

4.





4.

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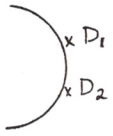
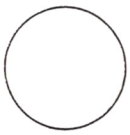
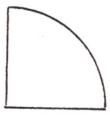
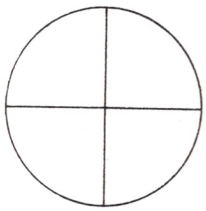
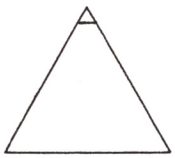
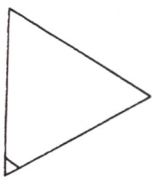
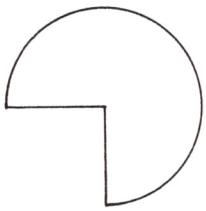
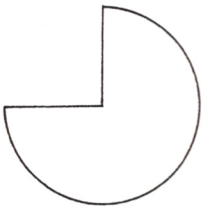
Part III
Rotation Exercises

Directions: Write the command and show the digitizes that would be needed to modify or construct the following exercises using the ROT command.

Before	After	Command Language
		# ROT COPY ANG180 : ENT D_1, DIG ORG D_2)
1. 		1.
2.		2.
3. 		3.
4. 		4.

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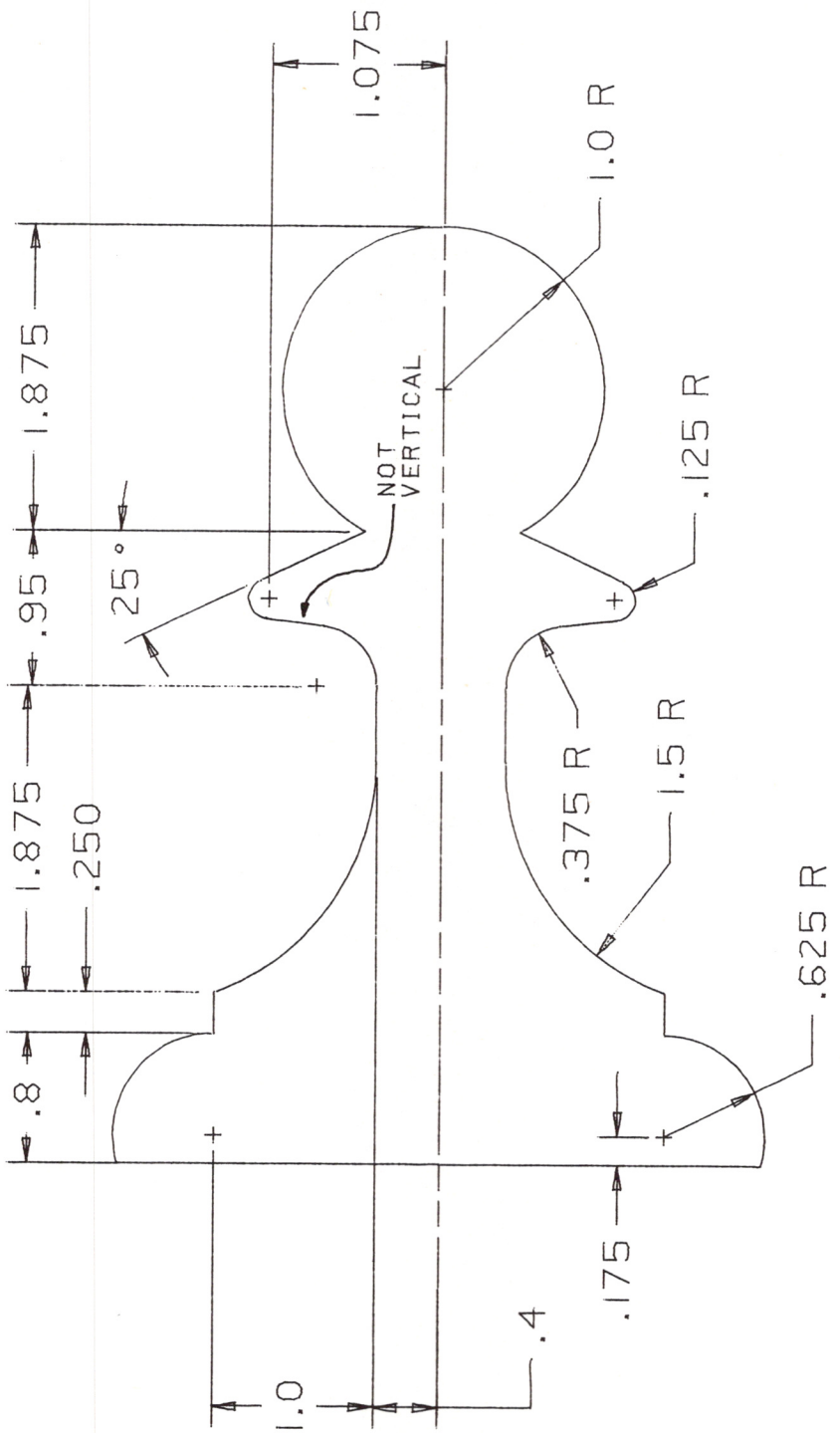
Unit 12 Lab Section

Exercise 12A



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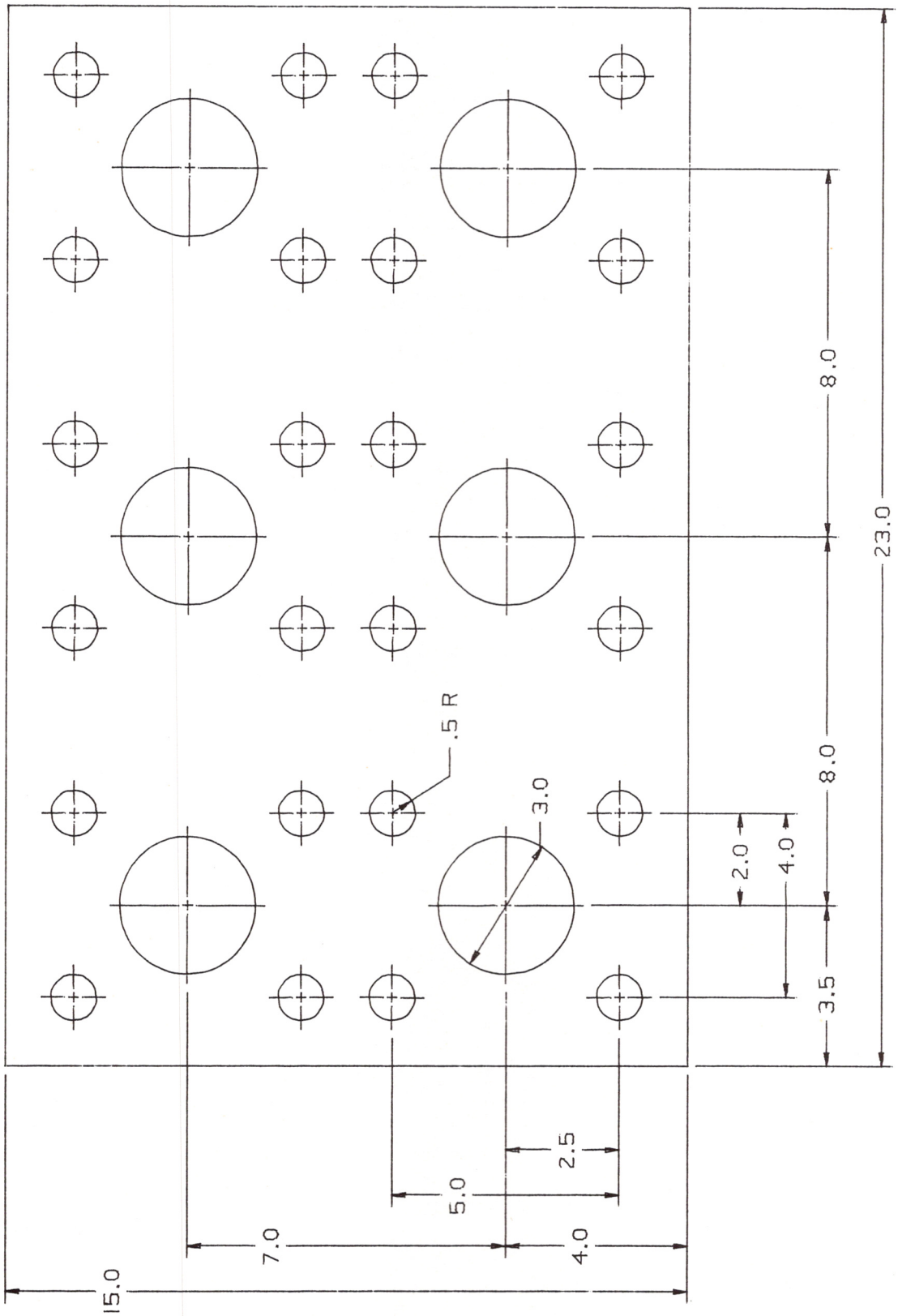
Exercise 12B



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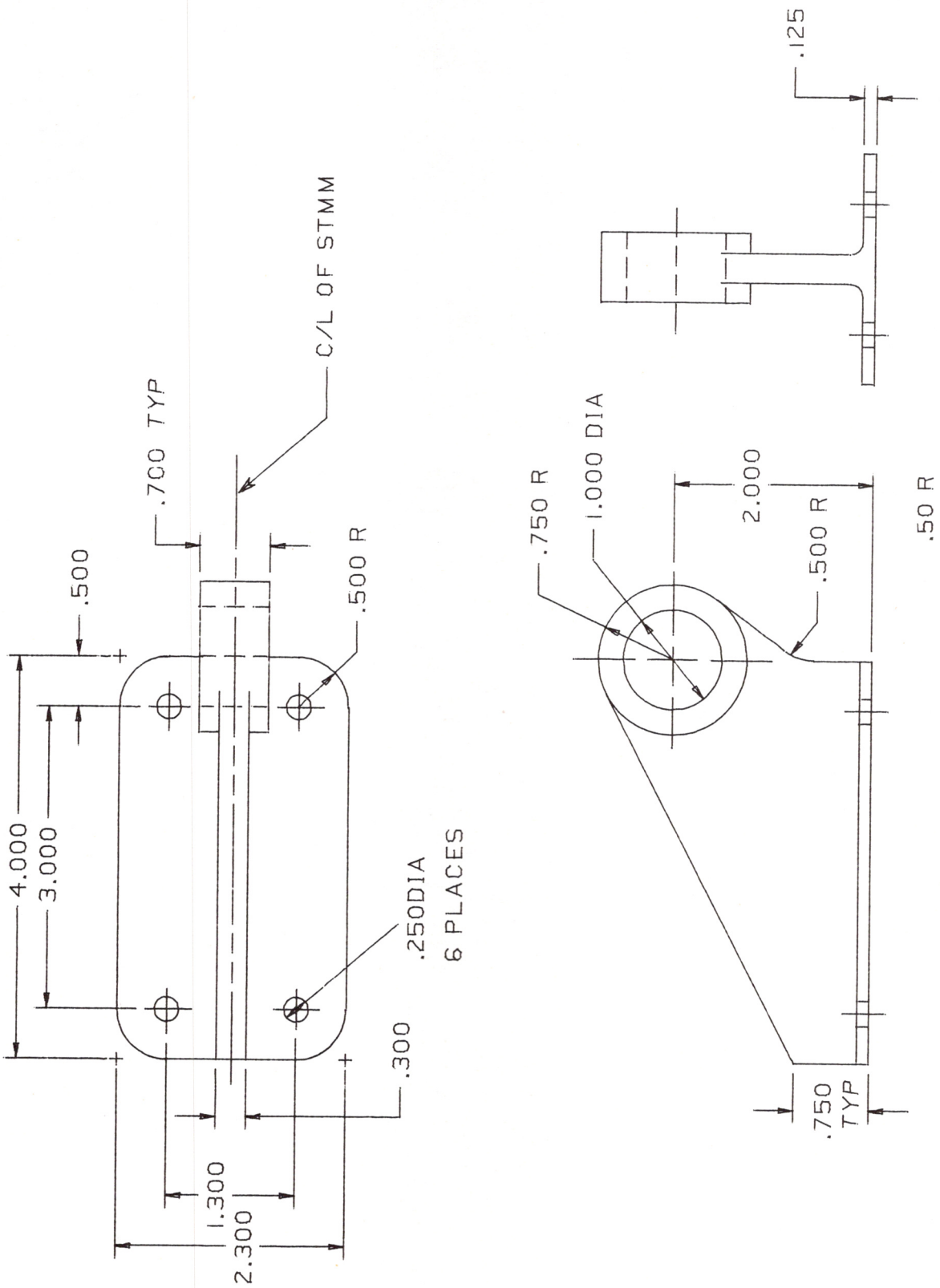
Exercise 12C



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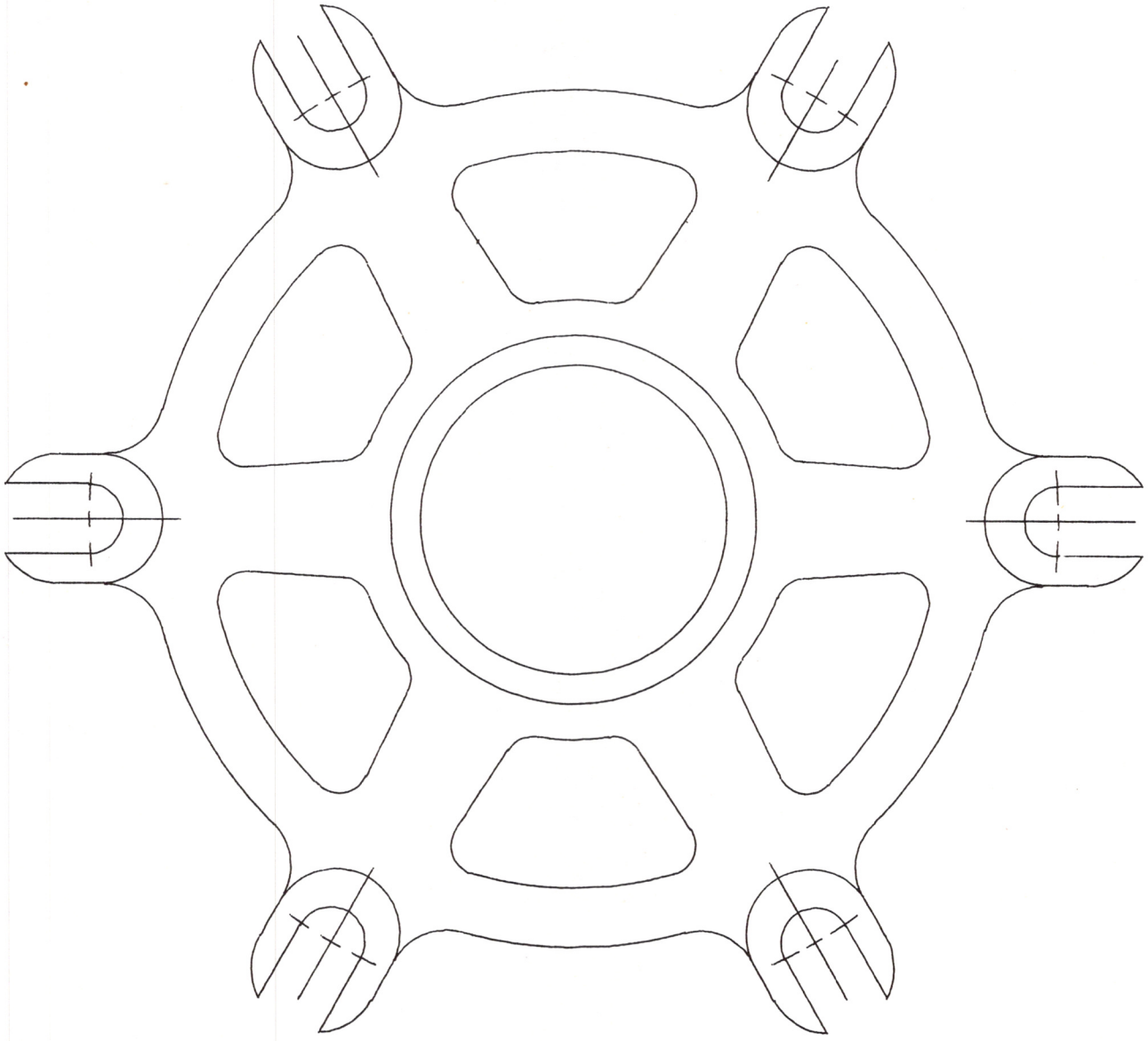
Exercise 12D

CREATE A PART SIMILIAR
TO THE DRAWING BELOW
MAKING USE OF THE ROTATE
COPY COMMAND.



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COMPUTER-AIDED DRAFTING

Unit 13
Layering Commands

MATCCAD™

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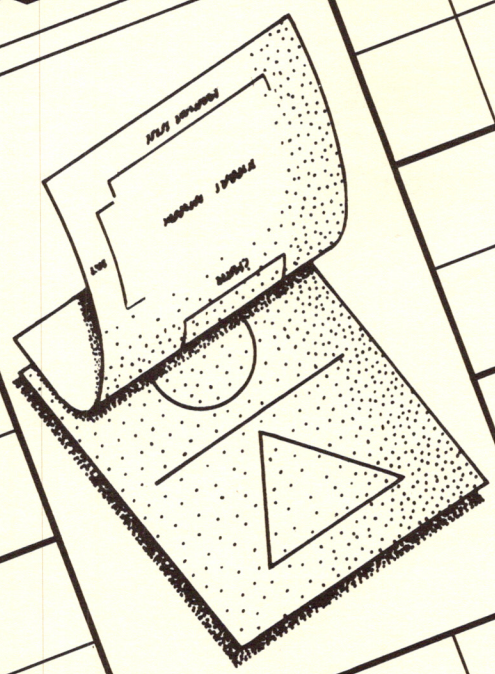

Introduction

The layering ability of a computer graphics system allows the operator to place different entity types or information on separate "transparent overlays" or layers. These layers can then be called up singly or in combination to present to the viewer only the information of interest on selected layers at a given time. Layering gets its widest use in fields such as cartography (mapping) where elevations, street names, lot lines, street lines, buildings, sewer, water, etc., can all be placed on distinct layers. In mechanical design applications, layering is used to separate metric dimensioning, notes, dimensions, etc.

This unit will give you instruction in the use of different layering commands as an aid in the construction and modification of two-dimensional drawings on the computer graphics system.

Objectives

At the completion of this unit you will be able to:

1. Select the layer of active construction using the SEL LAY command.

2. Make specified layers visible on the graphics display by the use of the ECH LAY command.

3. Use the LST LAY command to get information about the current layer status of a part, and the number of entities in your drawing.

4. Change entities from one layer to another using the CHG LAY command.

5. Construct a part while keeping the drawing, dimensions, and text on separate layers.

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Unit 13 Reference Section

Layering Commands

Layering is equivalent to having many transparent overlays lined up on top of one another with the ability to see or not to see any combination of the overlays at any given time. In the graphics system we are using there are 256 possible layers numbered 0-255 that can be used for the construction and display of a drawing. One, all or any combination of layers can be displayed at a given time. The construction layer is always visible and additional layers can be made visible as they are needed. In a typical application the basic part would be drawn on the default layer 0, with the other layers used for notes, dimensions, hidden features, etc.

Part I
The Select Layer Command

The Select Layer command specifies the layer number "n" to be used as the current construction layer.

SEL LAY n)

The construction layer is always visible whether it is echoed or not. Even though other layers may be echoed, you can only insert entities on the layer you have specified as the construction layer. Any entities moved, trimmed, rotated, mirrored, copied, or divided will remain on their original layer.

Part II
The Echo Layer Command

ECH LAY n)

The echo layer command makes visible those layers specified. Remember that the construction layer is always visible. A repaint is required whenever you change the echoed layers so that the newly selected layers will be displayed and the layers de-selected will be blanked.

Echo Layer Modifiers

Examples:

# ECH LAY 0 10 20 30)	This command will echo the layers 0, 10, 20, and 30.
# ECH LAY P15)	This will add layer 15 to the list of layers currently echoed.
# ECH LAY M6)	This will delete layer 6 from the list of layers currently echoed.
# ECH LAY ALL)	This will echo all layers 0 through 255.
# ECH LAY N)	This will echo no layers. Remember, however, that the construction layer is always visible.

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Part III
The List Layer Command

The **List Layer** command will indicate the number of the current construction layer and the numbers of the layers that are currently echoed.

LIST LAY)

The computer will respond by listing out a message similar to the one below:

WORK PORT LAYER STATUS : CONSTRUCTION LAYER : 2
ECHOED LAYERS : 2, 20, 21, 30 - 33

The **List Layer** Used command will tell you how many entities are in your part, and which layers have been used.

LIST LAY USED)

The computer will respond by listing out a message similar to the one below:

<LAYERS USED>

0 2 6

87 ENTITIES IN PART

Part IV
The Change Layer Command

The change layer command moves the selected entities from the layer they are currently on to the layer "n".

CHG LAY n : ENT D . . . D)

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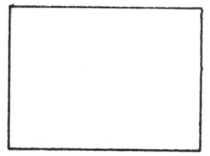
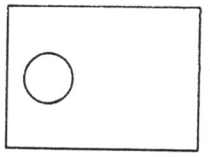
Part V
Layering Examples



SEL LAY 0) Layer 0 is selected as the construction layer (default).

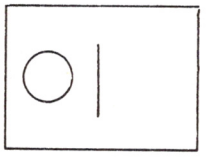
LIST LAY) (The LIST LAY commands that follow are optional.)

RPNT)



Construct a circle on the screen.

SEL LAY 1) Layer 1 is now the construction layer.

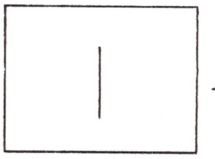


Construct a line on the screen.

LIST LAY)

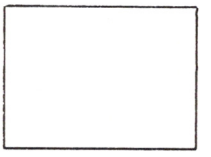
ECHO LAY 1)

LIST LAY)



RPNT)

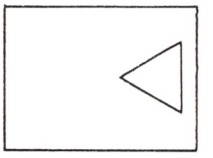
SEL LAY 2) Layer 2 is now construction layer.



ECHO LAY 2)

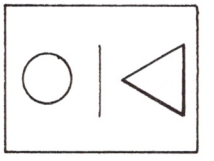
RPNT)

Construct a triangle on the screen.



ECH LAY ALL) All layers are visible (after a repaint).

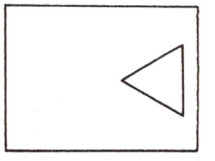
LIST LAY)



RPNT)

ECH LAY N) No layers are echoed, however the construction layer is always visible.

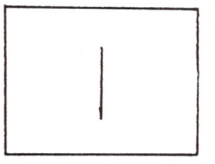
LIST LAY)



RPNT)

SEL LAY 1) Layer one is now the active layer and the visible layer (after a repaint).

LIST LAY)



RPNT)

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Unit 13 Exercise Section

Part I
Layering Exercises

Directions - Answer the following questions:

1. You are working on layer 2 and want to work on layer 0. What command would you use?

2. You want to add layer 10 to the list of layers currently echoed. What command would you use?

3. You are working on layer 0 and that's the only layer you want to see. What command would you use?

4. You are working on layer 7 and want to see layers 1-6 in addition. What command would you use?

5. You want to work on layer 0 and see all layers. What commands would you use?

6. What command would you use to find out which layer is the active layer and which layers are echoed?

7. If you want to know what layers are currently being used what command would you use?

8. If you want to know what layers are not used, what command would you use?

9. If you have some figures on layer 0 and you want the circles to be put on layer 6, what is the command you would use and where would you digitize?

10. What command will tell you how many entities are in your drawing?

11. What command would be used to move a line from where it is to layer 15?

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Unit 13 Lab Section

Lab Exercise 13

1. Call up exercise _____which you have previously created but have not yet dimensioned.
2. Dimension that exercise on layer 10.
3. Add text to the drawing of layer 11.
4. Move the circles to layer 1.
5. File the completed drawing.

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COMPUTER-AIDED DRAFTING

Appendix A
Other Graphics Systems

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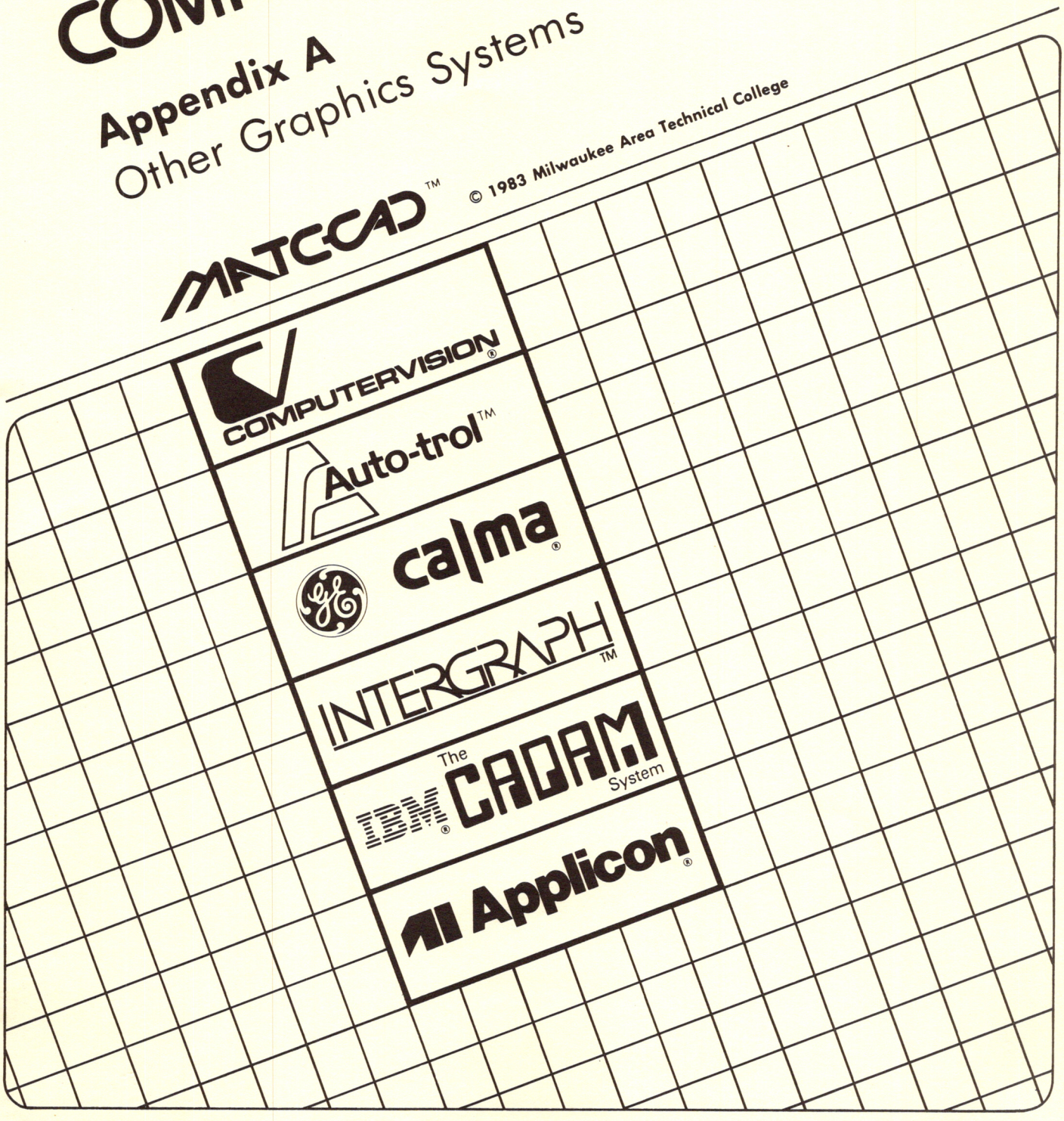

Computervision CADDS 4 System

The transition from working in Cadds 3 to Cadds 4 requires a minimal amount of time, especially when working in 2-D. The Computervision Cadds 4 graphics command language has the same basic structure as in Cadds 3. Each command consists of a verb, an optional noun, optional modifiers, a colon to separate the command from the data input section, and the data input part of the command.

The mnemonics for keywords (command abbreviations) in Cadds 4 consist of the first three or four letters of the word as it is spelled in conventional English. The punctuation has been standardized for all commands.

A major difference between Cadds 3 and Cadds 4 is the distinction made in Cadds 4 between model entities and drawing entities. Entities which are part of the model being created (lines, circles, arcs, etc.) are entered as model entities. Those which aid in describing and detailing the model (dimensions, borders, notes, etc.) are entered as drawing entities. Each Cadds 4 part consists of the database for the model and as many drawings of the model as desired. In order to view the model in any of the drawings the user must define a view.

The Cadds 4 Design Station has a graphics tablet with 427 programmable function keys on the tablet menu, compared with 220 keys on the Cadds 3 tablet menu. The Cadds 4 stylus has three buttons; one for digitizing, one for continuous digitizing, and one for rubber banding which is an added feature of Cadds 4. Another enhancement of the Cadds 4 Design Station is the Image Control Unit which provides for dynamic scrolling, zooming, and rotating.

Some examples showing typical commands in Computervision Cadds 4 follow:

DATA INPUT	Computervision CADDS 4
Prompts and Masks	
Prompt for digitized location	MODEL/DRAW loc
Prompt for entity selection	MODEL/DRAW ent
Mask for end of existing entity	End
Mask to limit selection to lines	Line
Mask to limit selection to circles	Cir

COMMANDS

Part Entry

Enter CAD Software	5>CADDS
Activate Part and Drawing	#5# ACTivate PARt MATC.SAMPLE.0 #5# ACTivate DRAWing ONE #5# DEF VIEW TOP: DRAW loc d DRAW loc d DRAW loc d #5# ECH FRAme

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Grids

Select a grid with .5 spacing	#5#SEL GRD G.5
Make grid visible	#5#ECH GRI ON
Make grid active	#5#ECH GRI SNAP

Geometry Creation

Lines between digitized locations	#5#INS LIN: MODEL loc ddd; MODEL loc dddd
Lines using coordinates	#5#INS LIN: MODEL loc XOYO,IX4,IY4,X5Y5,IY-7
Vertical line of 5" length	#5#INS LIN VER LENGTH 5: MODEL loc dd
Parallel line at a distance of 1"	#5#INS LIN PAR 1: MODEL ent d MODEL loc ddd
Perpendicular line of 3" length	#5#INS LIN PERP LENGTH 3: MODEL ent d MODEL loc dd
Circle with 2" radius	#5#INS CIR RAD 2: MODEL loc d
Circle with 2" diameter	#5#INS CIR DIAM 2: MODEL loc d
Fillet of .75" radius between 2 entities	#5#INS FIL RAD .75: MODEL ent dd

Text

Text with specified height, width, & slant	#5#INS TEX 'MATCCAD' HGT .3 WDT .2 SLT 30: MODEL loc d

Dimensioning

Horizontal Dimension	#5#INS LDim: DRAW/MODEL ent-end dd DRAW loc d
Vertical Dimension with 2 digit precision	#5#INS LDim VER PREc 2 : DRAW/ MODEL ent-end dd DRAW loc d
Diameter Dimension with 2 digit precision	#5#INS DDIm PREc 2 : DRAW/MODEL ent d DRAW loc d

Layering

Specify layer 1 as the construction layer	#5#SEL LAY 1
Make layer 1-5 and 10 visible	#5#ECH LAY 1-5
Add layers 5, 3, 100 to visible layer list	#5#ECH LAY INCL 5 3 100
Delete layer 3 from visible layer list	#5#ECH LAY EXCL 3
Move selected entity to layer 10	#5#CHA LAY 10: MODEL ent d
Make copy of entity on layer 10	#5#COPY ENT TO 10: MODEL ent d

Geometry Modification

Erase entity	#5#DEL ENT: MODEL ent d
Move entity	#5#TRAns ENT: MODEL ent d; MODEL loc dd
Move copy of entity	#5#TRAns ENT COPY: MODEL ent d; MODEL loc ORGdXOYO
Rotate entity 90°	#5#ROT COPY ENT ANG 90: MODEL ent d; MODEL loc POId

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Make copy of entity rotated 90°	#5#ROT ENT ANG 60: MODEL ent d; MODEL loc dIXON4M
Mirror copy of entity	#5#MIR ENT COPY : MODEL ent d; MODEL loc ENDdd
Change entity to dash line font	#5#CHA APP FONT DASH: DRAW/MODEL ent d

Screen Control

Increase image size on screen	#5#ZOOm DRAW IN
Decrease image size on screen	#5#ZOOm DRAW OUT
Increase image size on screen by 2.5 times	#5#ZOOm DRAW R2.5
Decrease image size by 2.5 times	#5#ZOOm DRAW R.4
Fill screen with selected rectangular area	#5#ZOOM DRAW WIN : Model loc d d
Fill screen edge to edge with drawing	#5#ZOOm DRAW ALL
Shift screen image on screen	#5#SCRoll VIEW: view dd

Part Exit

File drawing	#5#EXIT PARt File
Exit from CAD software to system level	#5#EXIT CADds
Log-off of system	5>LOGOUT

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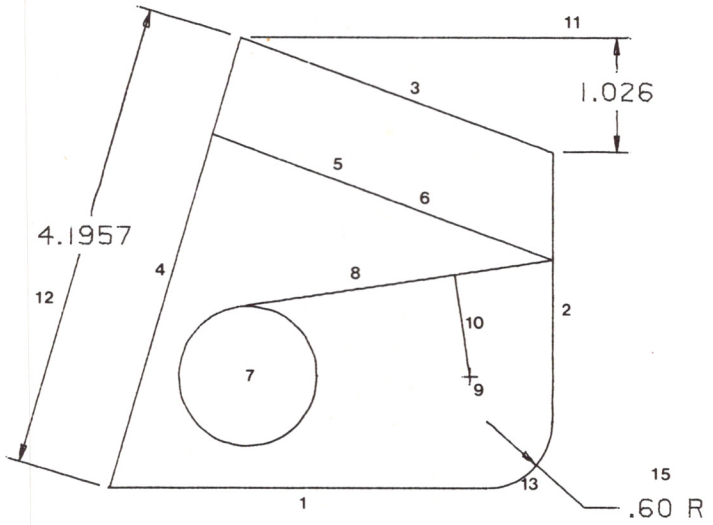
1. Horizontal line, 4 inches long with left end at X=0 Y=0
2. Vertical line 3 inches long starting at the end of line 1
3. Three inch long line at an angle of 160° from horizontal
4. Line from the end of line 1 to the end of line 3
5. Some line parallel to line 3 at a parallel distance of 0.9 inches
6. Trim line 5 at the intersection of lines 2 and 4
7. Circle of 1.25 inch diameter at position X=1.25 Y=1
8. Line tangent to circle 7 and going to the end of line 5
9. Point at location X=3.25 Y=1
10. Line perpendicular to line 8, from point 9 to line 8
11. Vertical dimension of line 3
12. True length dimension of line 4 with four decimal precision
13. Fillet of 0.6 inch radius between lines 1 and 2
14. Text containing the system name
15. Radial Dimension with two decimal precision



COMPUTERVISION CADDS4
14

```
#3#ACT PARt EXAMPLE
#3#ACT DRAw 1
#3#DEF VIEw TOP CPL TOP: DRAW loc d
#3#ECH FRAme
#3#INS LIN: MODEL loc XOYO,IX4,IY3,IR3A160
#3#INS LIN: MODEL loc END d d
#3#INS LIN PAR.9: MODEL ent d MODEL loc d d d
#3#TRIm ENT INTOF: MODEL ent d MODEL ent d MODEL ent d
#3#INS CIR DIAME 1.25: MODEL loc X1.25Y1
#3#INS LIN TANTO: MODEL ent d MODEL ent END d
#3#INS POInt: MODEL loc X3.25Y1
#3#INS LIN PERP: MODEL ent d MODEL loc POInt d END d
#3#INS LDIm VER : DRAW/MODEL ent-end d d DRAW loc d
#3#INSLDIM PPOint PREc 4: DRAW/MODEL ent-end d d DRAW loc d
#3#INS FIL RAD .6: MODEL ent d d
#3#INS TEXt /COMPUTERVISION CADDS4/: MODEL loc d
#3#EXIT PARt F
```

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